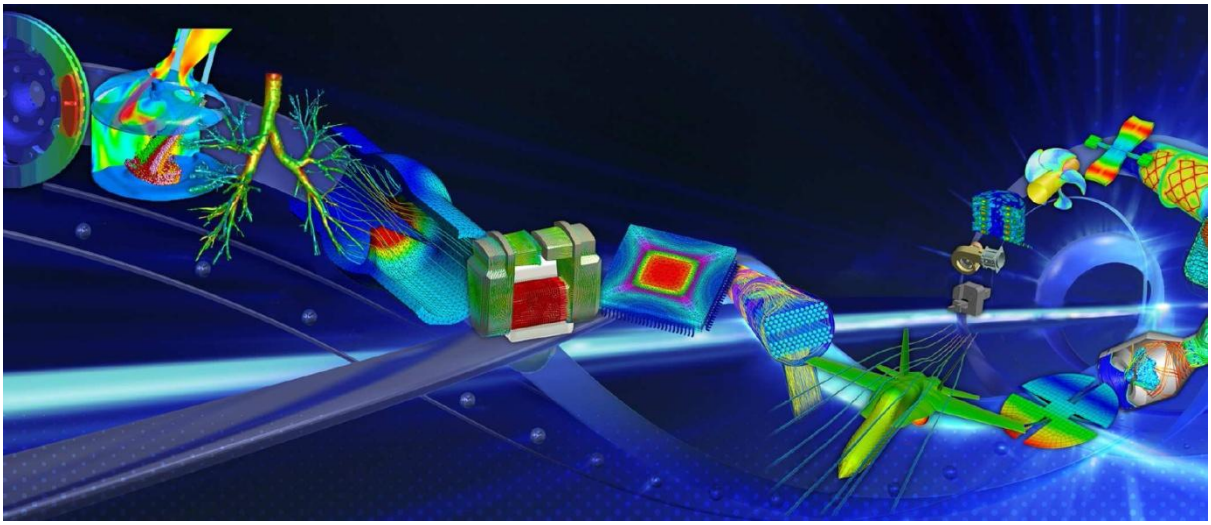


AUTODYN® User Manual Version 12.1

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 12.1
November 2009

ANSYS, Inc. is certified to ISO 9001:2008.

Copyright and Trademark Information

© 2009 SAS IP, Inc. All rights reserved. Unauthorized use, distribution, or duplication is prohibited.

ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.
ANSYS, Inc. is certified to ISO 9001:2008

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non DOD licenses).

Third Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Table of Contents

AUTODYN User Manual 1
Overview 1
Overview 1
Overview - Tool Bar 3
Overview - Navigation Bar 5
Overview - Dialog Panel and Dialog Windows 7
Overview - Command Line Panel 12
Overview - Undock Command Line 16
Overview - Command File Window 18
Overview - Message Panel 20
Overview - HTML review windows 21
Overview - File names 22
Overview - On-Line help 23
Overview - Supported Operating Systems and Compilers 24
Overview - Initialization Settings 25
Slide Viewer 30
Overview - AUTODYN in Workbench 35
Overview - AUTODYN solver options in Workbench 44
Pull-down Menus 45
Pull-down Menus 45
File 47
Pull-Down Menu-File 47
Pull-down Menu - File - New 50
Manage Folders 53
Pull-down Menu - File - Manage Folders 53
Manage Folders - New Folder 54
Pull-down Menu - File - Save As 55
PullDown Menus - File - SaveProject 56
Pull-down Menu - File - Save As Version 57
PullDownMenus - File - Export To Version 58
Pull-down Menu - File - Export PostScript 59
Pull-down Menu - File - Export VRML 66
Import 68
Pull-down Menu - Import 68
Pull-down Menu - Import - TrueGrid 70
Pull-down Menu - Import - ICEM 74

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Import - LS-DYNA 76
Pull-down Menu - Import - Nastran 80
Pull Down Menus - Import - Convert IJK to Unstructured 83
Setup 84
Pull-down Menu - Setup 84
Pull-down Menu - Setup - Description 86
Pull-down Menu - Setup - Symmetry 88
Execution 89
Pull-down Menu - Execution 89
Pull-down Menu - Execute - Batch Runner 90
View 92
Pull-down Menu - View 92
Pull-down Menu - View - Slideshow - Setup 94
Pull-down Menu - View - Performance Profiler 96
Options 98
Pull-down Menu - Options 98
Pull-down Menu - Options - Interactivity 99
Pull-down Menu - Options - Clipping Planes 100
Pull-down Menu - Options - Default Settings 101
Pull-down Menu - Options - Pause License 102
Pull-down Menu - Options - Create User-Subroutine Project 103
Pull-down Menu - Help 104
Plots 105
Plots 105
Plots - Select Contour Variable 107
Plots - View range and slices (Structured) 108
View range and slices (SPH/Unstructured) 109
Settings 110
Settings 110
Settings - Display 112
Settings - Grid 119
Settings - Legend 120
Settings - Node/Element Numbers 122
Materials 123
Settings - Materials 123
Settings - Materials - Modify Material Color 125
Contour 126

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Contour 126
Settings - Contour - Profile Plot 129
Settings - Velocity vector 131
Settings - Gauge point 133
Settings - Boundary 135
Settings - Joins 136
Settings - Axes 137
Settings - Detonation 139
Settings - Eroded Nodes 140
Settings - Unused Regions 141
Settings - Polygons 143
Settings - Beam Sections 144
Settings - Shell Thickness 149
Settings - Material Direction 150
Settings - Marker 152
Settings - Shell Normal 155
Settings-Part Highlights 156
History 158
History 158
History - Single Variable Plot 161
History - Reduce 163
History Write Text Output 164
Multiple Variable Plot 165
History - Multiple Variable Plot 165
History - Multiple Variable Plot - Set All Plots 167
History - Multiple Variable Plot - Modify Selected Plot 169
History - Min/Max 171
History - Set Scales 172
History - Set Plot Styles 174
History - Set Legend Position 176
History - Load 177
Slides 179
View Slides (Slide Viewer) 183
Materials 188
Materials 188
Materials - New / Modify 190
Materials - Copy 195

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Materials - Delete 196
Materials - Load 197
Initial Conditions 199
Initial Conditions 199
Initial Conditions - New/Modify 201
Initial Explicit Model Initialization 204
Initial Conditions - Delete 207
Boundaries 208
Boundaries 208
Boundaries - New / Modify 209
Boundaries - Delete 211
Parts 212
Parts 212
New 218
Parts - New 218
Parts - New - Select Predef (2D) 221
Parts - New - Select Predef (3D) 229
Parts - New - Select Predef (3D Shell) 238
Parts - New - Define Zoning (2D) 241
Parts - New - Define Zoning (3D) 251
Parts - New - Define Zoning (3D Shell) 261
Parts - New - Fill Part (2D) 265
Parts - New - Fill Part 267
Parts - New - Fill Part (Shell) 270
Parts - Copy 272
Parts - Delete 273
Parts - IJK Range 274
Parts - Activation 275
Parts - Rename 276
Parts - Print 277
Parts-Pack(Fill) 278
Zoning 279
Parts - Zoning 279
Parts - Zoning - Box 282
Parts - Zoning - Cylinder 284
Parts - Zoning - Sphere 286
Parts - Zoning - Ogive 288

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Quad 290
Parts - Zoning - Hex 292
Parts - Zoning - Frag/Bric (page 1) 294
Parts - Zoning - Frag/Bric (page 2) 296
Parts - Zoning - Transition 298
Parts - Zoning - Node 299
Parts - Zoning - Line 300
Parts - Zoning - Surface 302
Parts - Zoning - Volume 304
Parts - Zoning - Translate 306
Parts - Zoning - Rotate 307
Parts - Zoning - Scale 308
Parts - Zoning - Copy 309
Parts - Zoning - Delete 310
Parts - Zoning - Save 311
Parts - Zoning - Restore 312
Parts - Zoning - Refine 313
Parts-Zoning-Fill Block 315
Parts-Zoning-Fill Block 315
Parts-Zoning-Fill Block-Implicit 316
Parts-Zoning-Fill Block-Explicit 318
Fill 320
Parts - Fill 320
Parts - Fill - Block 322
Parts - Fill - I / J / K Plane 324
Parts - Fill - Node 326
Parts - Fill - Unused 328
Composite 329
Parts - Fill - Composite 329
Parts - Fill - Multi-Fill 333
Parts - Fill - Datafile 335
Parts - Fill - Part Fill 340
Parts - Fill - Block Remap 342
Boundary 347
Parts - Boundary 347
Parts - Boundary - Block 348
Parts - Boundary - I / J / K Plane 349

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Boundary - Node 350
Parts - Boundary - Clear 351
Parts - Boundary - Apply Interactively 352
Parts - Boundary - Clear Interactively 353
Gauges 354
Parts - Gauges 354
Parts - Gauges - Add 356
Parts - Gauges - Move 357
Parts - Gauges - Delete 358
Parts - Gauges - Add Unstructured 359
Parts - Gauges - Move Unstructured 360
Parts - Gauges - Delete Unstructured 361
Solvers 362
Parts - Solvers (Structured) 362
Parts - Solvers (Structured) - Type 363
Parts - Solvers (Structured) - Solution (2D Shells) 364
Parts - Solvers (Structured) - Solution (3D Shells) 365
Parts - Solvers (Structured) - Solutions (Beams) 366
Parts - Solvers (Structured) - Solution (3D Euler-FCT) 367
Parts - Solvers (Structured) - Solution (2D SPH) 368
Parts - Solvers (Structured) - Solution (3D SPH) 370
Parts - Solvers (Structured) - Dezone-(2D & 3D Euler) 372
Parts-Solver (Structured) - Dezone (2D & 3D Lagrange/ALE) 373
Parts - Solvers (Structured) - Jetting 375
Parts - Solvers (Unstructured) - Solid 377
Parts - Solvers (Unstructured) - Shell 380
Parts - Solvers (Unstructured) - Beam 382
Motions 384
Parts - Motions 384
Parts - Motions - Block 385
Parts - Motions - I / J / K Plane 386
Parts - Motions - Node 387
Parts - Motions - Modify 388
Geometry 389
Parts-Geometry - SPH 389
Parts - Geometry - New/Modify 391
Parts-Geometry-Copy 394

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts-Geometry-Import Part 395
Parts-Geometry-Translate 398
Parts-Geometry-Rotate 399
Components 400
Components 400
Components - New 402
Components - Modify 404
Components - Delete 406
Components - Material 407
Components - Velocity 408
Components - Initial Conditions 409
Components - Apply Boundary 410
Components - Clear Boundary 411
Components - Translate 412
Components - Rotate 413
Components - Scale 414
Components - Copy 415
Groups 416
Groups 416
Groups - New 427
Groups - Rename 428
Groups - Copy 429
Groups - Delete 430
Groups - Apply 431
Groups - Clear 432
Groups - Material 433
Groups-Velocity 434
Groups - Initial Condition 435
Groups-Translate 437
Groups-Rotate 438
Groups-Scale 439
Joins 440
Joins 440
Joins - Join 446
Joins - Unjoin 448
Joins - Matrix 449
Merge Joined Nodes 450

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Bonded Face Connections 451
Interactions 454
Interactions 454
Lagrange/Lagrange 455
Interactions - Lagrange / Lagrange 455
Lagrange/Lagrange Introduction 457
Interactions - Lagrange / Lagrange - Create 461
Interactions - Lagrange / Lagrange - Add 462
Interactions - Lagrange / Lagrange - Remove 464
Interactions - Lagrange / Lagrange - Matrix 466
Interactions - Lagrange / Lagrange - Range 468
Interactions - Lagrange / Lagrange - Friction 469
Lagrange/Lagrange - 2D Gap Contact Options 472
Interactions-Lagrange/Lagrange-2D Trajectory Contact Options 476
Interactions-Lagrange/Lagrange-3D Gap Contact Options 480
Interactions-Lagrange/Lagrange-3D Trajectory Contact Options 487
Euler/Lagrange (2D) 494
Interactions - Euler / Lagrange (2D) 494
Interactions - Euler / Lagrange Introduction 499
Interactions - Euler/Lagrange (2D) - New Polygon 502
Interactions - Euler/Lagrange (2D) - Velocity 503
Interactions - Euler/Lagrange (2D) - Porosity 504
Interactions - Euler/Lagrange (2D) - New Interaction 505
Euler/Lagrange (3D) 506
Interactions - Euler - Lagrange (3D) 506
Interactions - Euler / Lagrange (3D) - Select 508
Interactions - Euler / Lagrange (3D) - Thickness 510
Interactions - Euler / Lagrange (3D) - Joins 512
Interactions - Euler / Lagrange (3D) - Weak Coupling 514
Detonation (2D) 517
Detonation (2D) 517
Detonation (2D) - Point 519
Detonation (2D) - Line 521
Detonation (2D) - Circle 523
Detonation (2D) - Manual 525
Detonation (3D) 526
Detonation (3D) 526

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (3D) - Point 528
Detonation (3D) - Plane 530
Detonation (3D) - Cylinder 532
Detonation (3D) - Sphere 534
Detonation (3D) - Manual 536
Parallel 537
Parallel 537
Parallel - Add / Modify 540
Parallel - Copy 542
Parallel - Sub-domains 543
Parallel - Task Assignment 545
Parallel Automatic Decomposition 549
Automatic decomposition of Euler-FCT parts 551
Parallel - Decomposition of beam parts 555
Output 557
Output 557
Output-Part Summary Variables 564
Outputs-Results Files 567
User Variables 571
User Variables 571
User Variables - Add/Modify 572
Controls 574
Controls 574
Controls-Mass scaling 584
Controls - Global Erosion 587

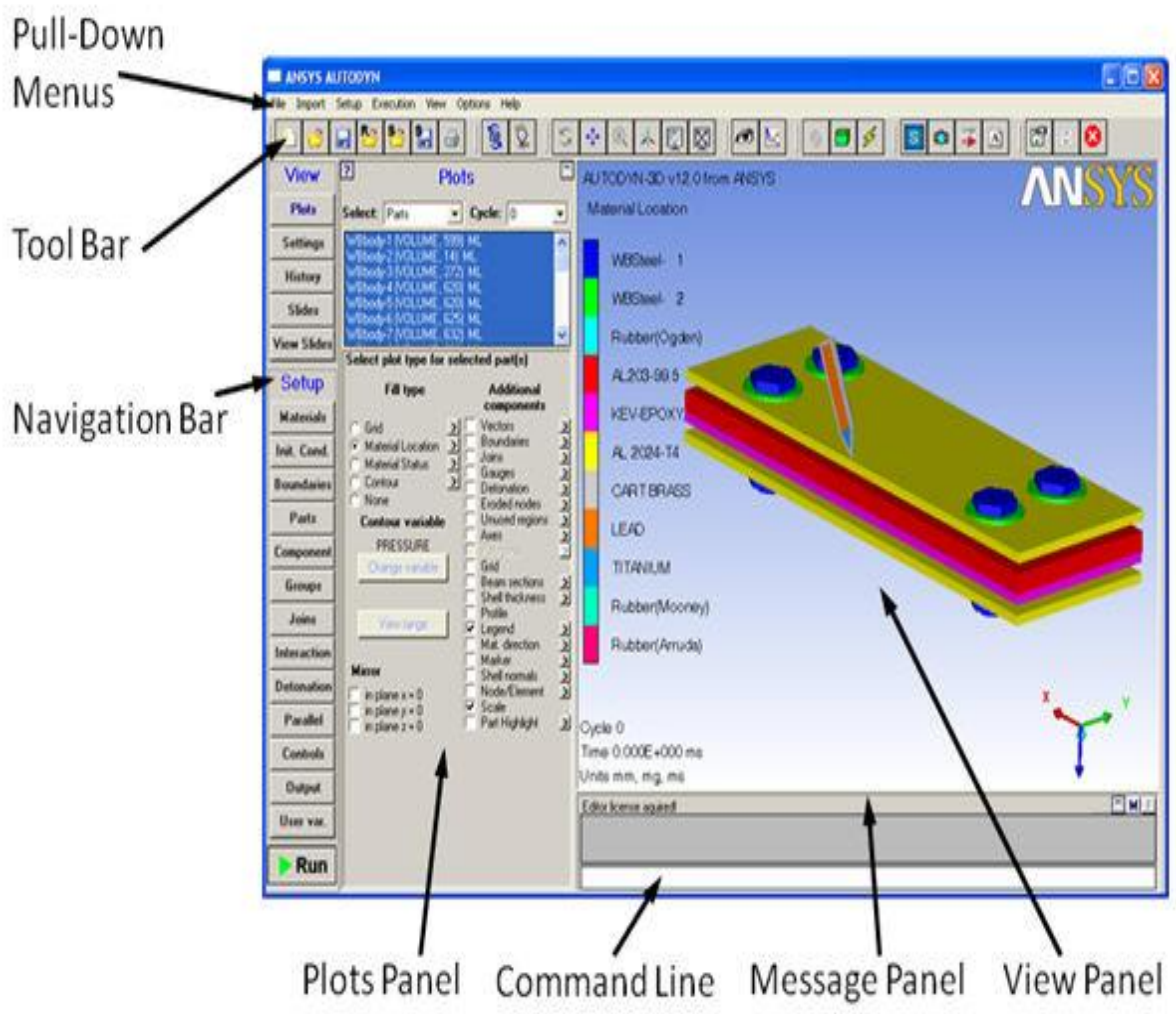
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

AUTODYN User Manual

Overview

Overview

ANSYS AUTODYN is an explicit analysis tool for modeling nonlinear dynamics of solids, fluids, gas, and their interaction, and forms part of the ANSYS Workbench suite.



ANSYS AUTODYN provides a user-friendly graphical user interface (GUI), combining pre-processor, solver and post-processor in the same windows environment. A single executable performs simulations in both two and three dimensions.

The GUI is laid out with buttons running horizontally along the top and vertically down the left hand side of the main window. The horizontal group of buttons is

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

referred to as the **Tool Bar** and the vertical group as the **Navigation Bar**. The Tool Bar and Navigation Bar offer quick access to options also available through the **Pull-down Menus**.

The main window is composed of a number of **Panels** (View, Dialog, Message and Command Line).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Tool Bar

The Tool Bar (and the Navigation Bar) provides shortcuts to options available in the pull-down menus.

The Tool Bar buttons and their use are described below.

Create a new model (this functionality is only available when AUTODYN is started up outside ANSYS Workbench)

Open an existing model (this functionality is only available when AUTODYN is started up outside ANSYS Workbench)

Save current model to current filename (this functionality is only available when AUTODYN is started up outside ANSYS Workbench)

Open a results file

Open a settings file

Save current plot view parameters to a settings file

Print

Transform object – default to on

Transform light source

Rotate model

Translate model

Zoom function

Set view – brings up new window so that exact view can be set

Reset model

Fit model to view panel

Examine model

Profile window

Toggle wireframe mode (on/off)

Perspective on/off

Hardware acceleration on/off

Setup slide show

Create image of current view

Record slideshow

Create textslide

Show/hide Navigation Bar

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Manual/Automatic refresh

Refresh screen

Stop all plotting

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Navigation Bar

The Navigation Bar contains two groups of buttons.

The top group of buttons (**View**) let you control what you see in the View Panel. From here you can access and modify plot settings, view history plots, and create or view slide shows / animations :

The lower group of buttons (**Setup**) let you access dialog panels to set up your model. Starting at the top button (**Materials**) and working down to the **Run** button, you can build your model in the most efficient and logical way :

The Problem Setup section explains this procedure in detail:

The Run button at the foot of the bar is used to run (execute) problems :

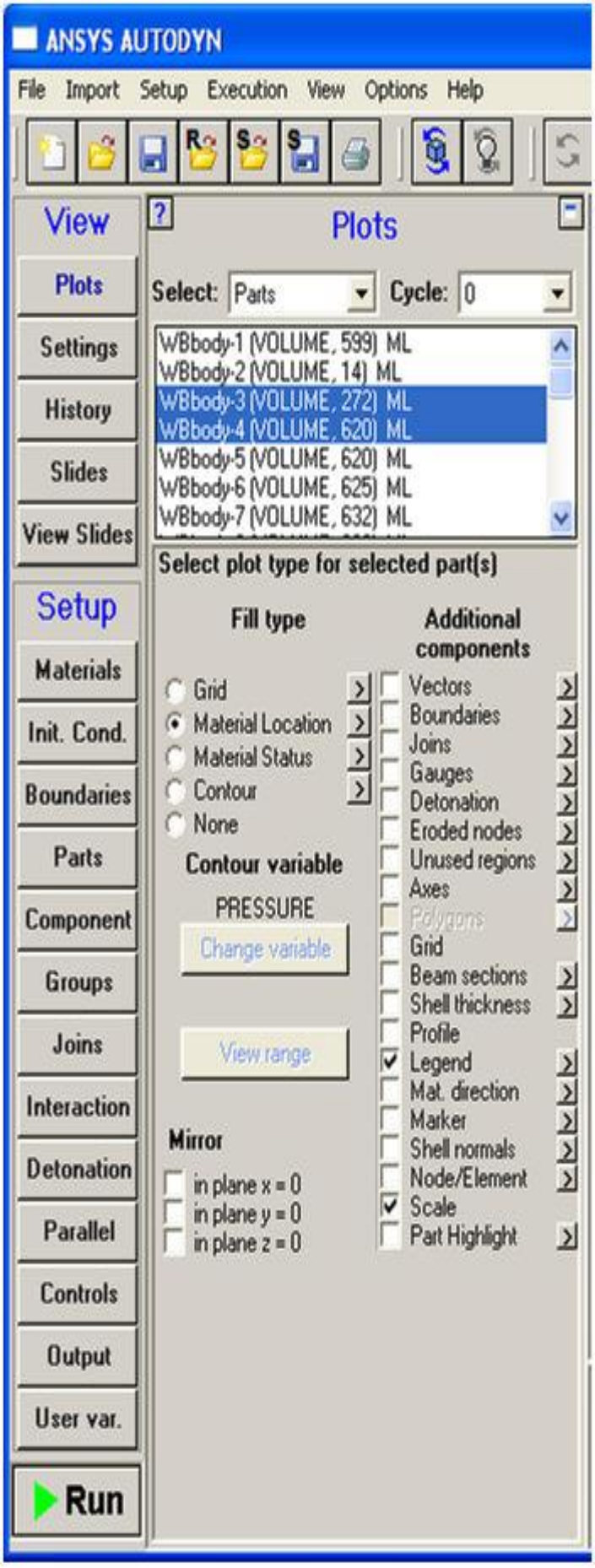
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Dialog Panel and Dialog Windows

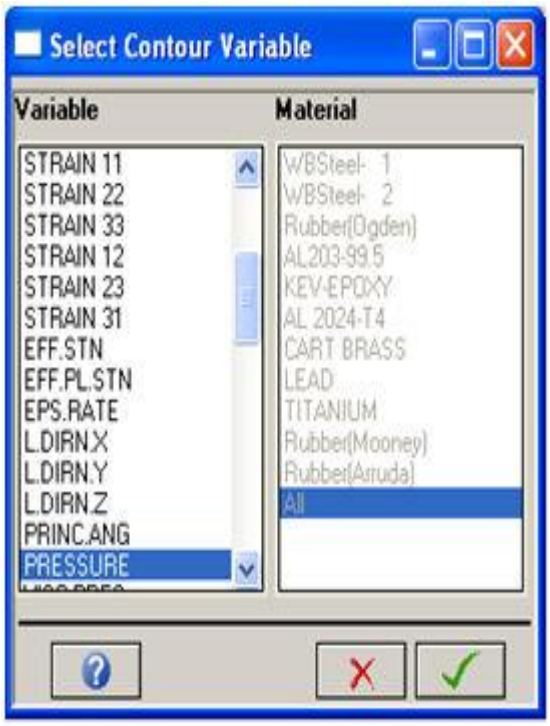
When you select an option from the Navigation Bar, dialog for that option appears in the **Dialog panel :**

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The Dialog panel generally contains input fields and buttons that access further input options. Pressing a button in the dialog panel will display further dialog either within the dialog panel or in a new **Dialog Window** that is opened :



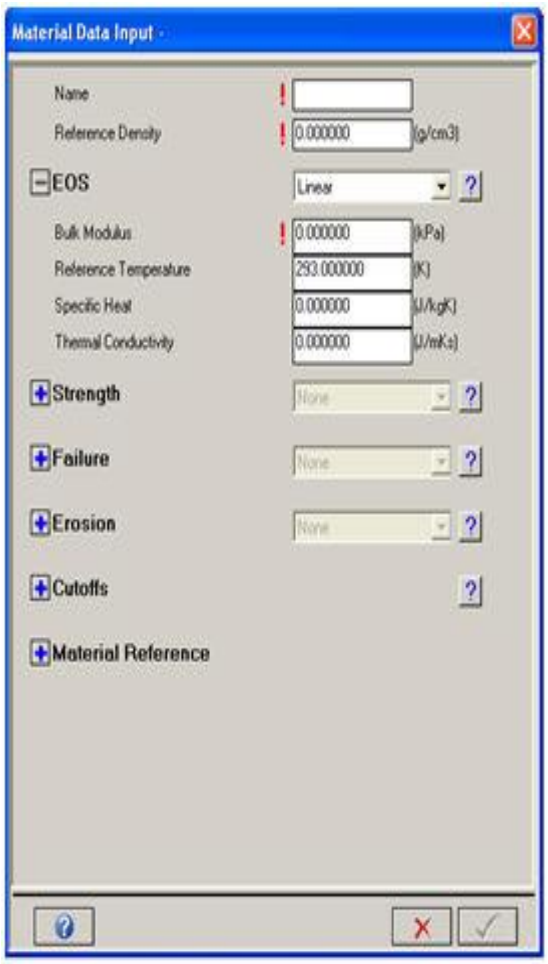
At the bottom of all Dialog Windows there are three buttons.

The button with a question mark provides help on the current windows function.

The two other buttons are (Cancel) and (OK). Pressing the Cancel button will close the window, without applying any of the changes that you made within it. The OK button closes the window and applies your changes.

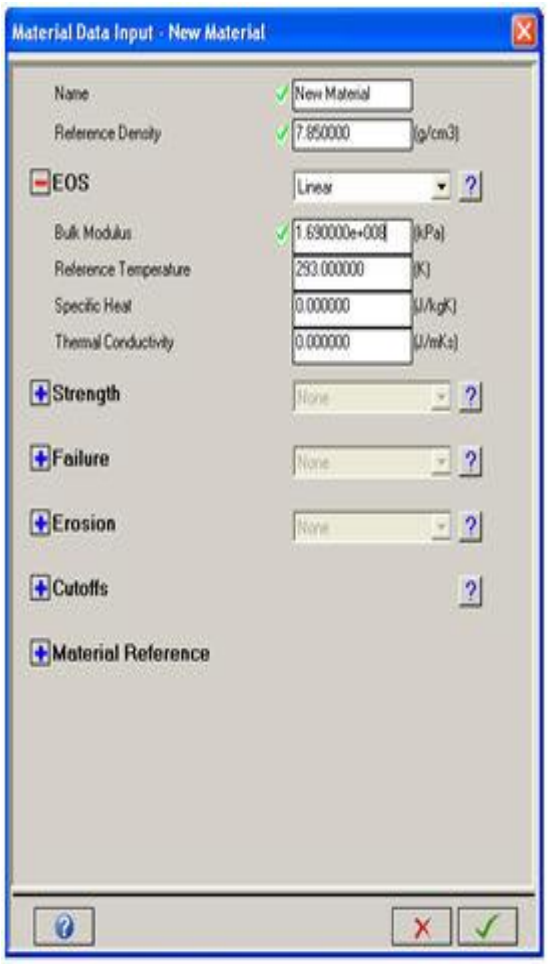
In certain cases, an **Apply** button will be present. This provides the ability to apply any changes made within the window without closing the window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



In the dialog panel or dialog windows, input data that is required for a task to complete is marked ! (above). When you enter an appropriate value, the mark changes to ✓ (below). The OK button will remain inactive until you provide appropriate values for all required fields :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Command Line Panel

The command language will provide the user with the capability to

- Record operations made in the GUI
- Modify/Playback/repeat operations
- Load operations previously saved to file
- Setup models using the command language

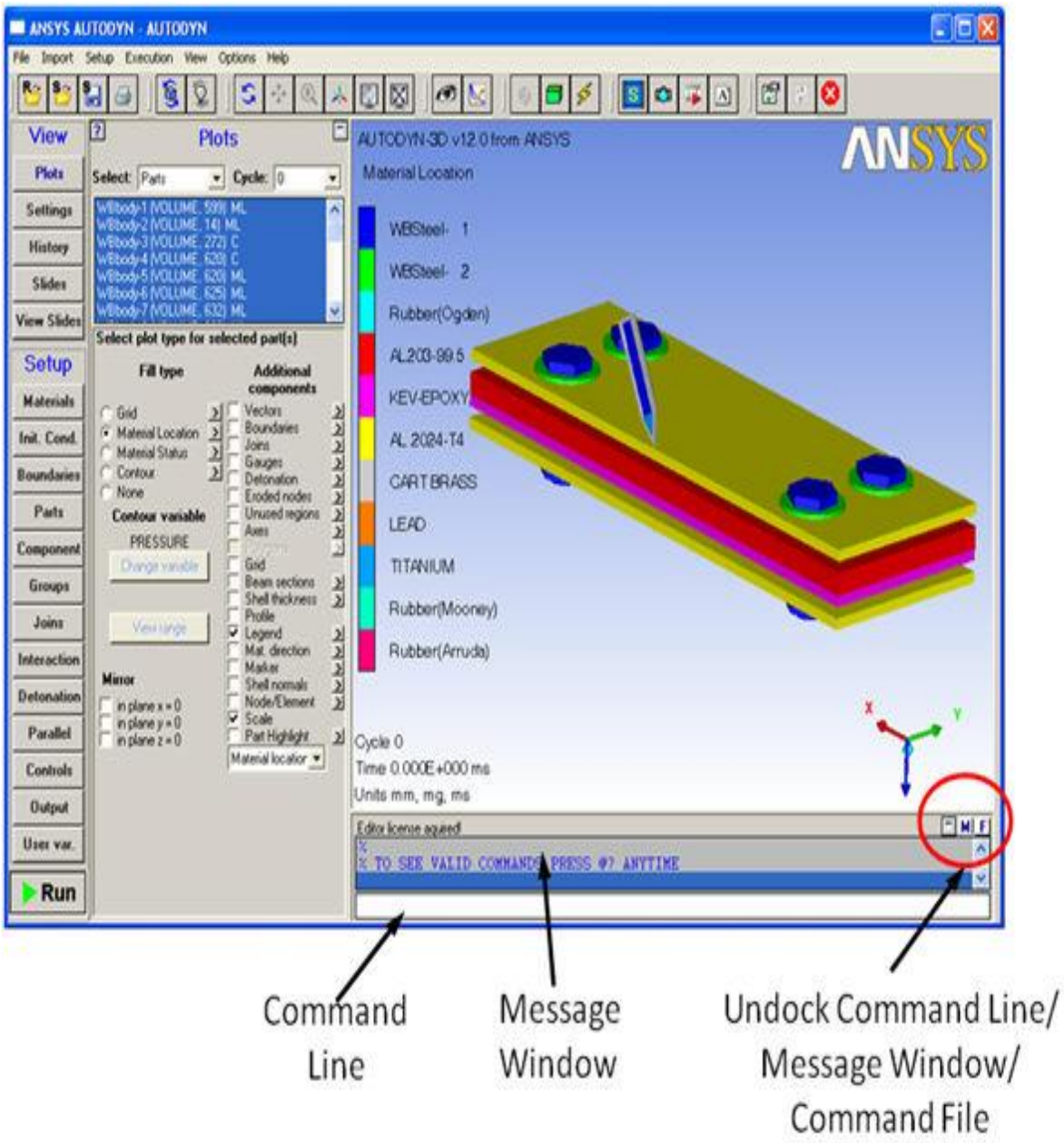
This feature has Beta status and must be activated by the user by adding the following lines above the Additional User Title option in the autodyn.ini file:

command_line_on
1

By activating this flag you accept that this is a Beta option and therefore may not be as stable or robust as expected in a general release version.

The command line is initially inactive until a model is either loaded or created. To the right of the message line there are 2 buttons in addition to the Message window button that are related to the command line.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Commands can be typed directly into the command line with multiple commands being concatenated by using a ';' between the commands, i.e. PARTS:;TARGET to select a part named TARGET. Useful/help information is output to the Message/Output window. Note however that the command language is more easily used when the command window is undocked from the main user interface.

The command line creates two command files:

1. autodyn.adc - this is located in the directory C:\Documents and Settings\username\Local Settings\Temp and is overwritten every time a new AUTODYN session is started.
2. ident.adc - this file is created every time a new model is created. If the model is subsequently modified in a different AUTODYN session the new commands are appended to this file. Therefore the ident.adc should contain all commands that have been executed on the model since it was first created.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

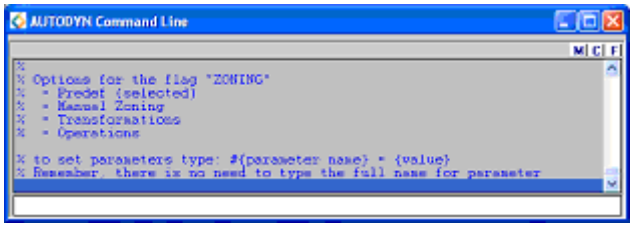
A typical series of commands will look similar to that shown below:

```
PARTS:
target
 @ZONING
  #OPERATION TYPE      = Zoning
   #ZONING             = Predef
    #PREDEF GEOMETRY    = Box
     X Origin          =   0.0000000E+00
     Y Origin          =   0.0000000E+00
     Z Origin          =   0.0000000E+00
     DX                =   10.00000
     DY                =   10.00000
     DZ                =   10.00000
     #GRADING I         = NO
      #GRADING J        = NO
       #GRADING K       = NO
 @DO
 @FILL
  #OPERATION TYPE      = Fill
   #FILL BY INDEX      = Block
    I min              =         1
    I max              =        11
    J min              =         1
    J max              =        11
    K min              =         1
    K max              =        11
    #FILL WITH INITIAL CONDITION SET? = No
     #FILL MATERIAL     = AL 2024
      Density           =   2.785000
      Int. Energy       =   0.0000000E+00
      X Velocity        =   0.0000000E+00
      Y Velocity        =   0.0000000E+00
      Z Velocity        =   0.0000000E+00
      #RADIAL VELOCITY   = No
       #ANGULAR VELOCITY  = No
 @DO
```

The 'PARTS:' command word is the name of the current command structure. (When other parts of the interface are covered by the command line, each of the buttons on the Navigation panel will be a command structure, i.e. Materials, Boundaries etc.)

The name of the part currently selected is then given, in this case it is called 'target'. In this example the command being executed is a zoning command followed by a fill command- @ indicates a command. To see the current list of available commands '@?' can be typed at any time.

A # denotes an option. In the above example, #zoning allows you to choose from the list of zoning options. Typing '?#' will give you the list of options – for zoning the options are shown below:


AUTODYN Command Line
%
% Options for the flag "ZONING"
% = Predef (selected)
% = Manual Zoning
% = Transformations
% = Operations
% to set parameters type: #(parameter name) = (value)
% Remember, there is no need to type the full name for parameter

For the example given here predef is chosen as the option. The predef zoning option has a sub-option '#PREDEF GEOMETRY' which allows the user to choose the predef they want to zone. In this simple example the box predef is chosen and the required parameters have been set.

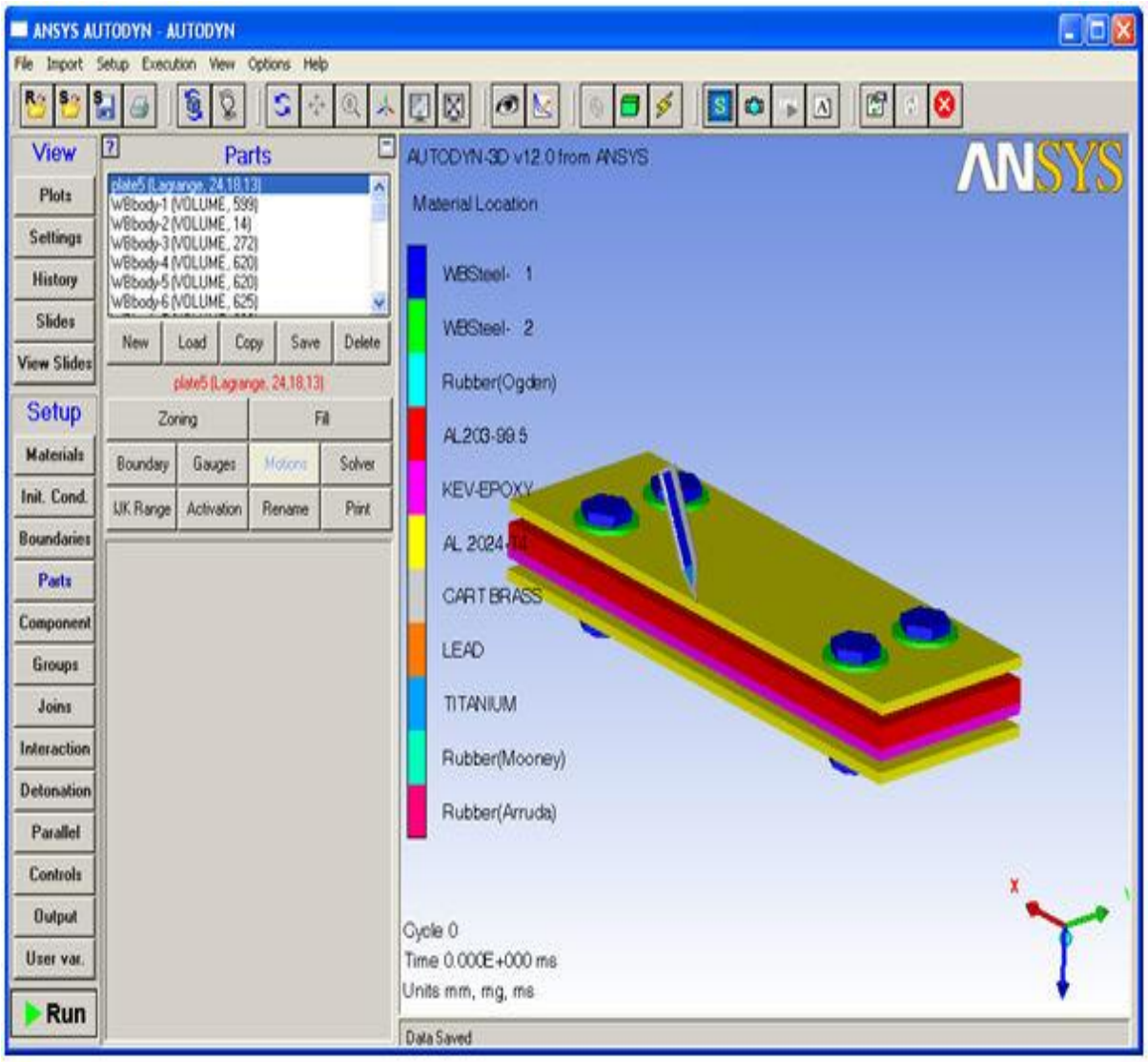
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

When all the parameters have been set the '@DO' command is issued which executes the command.

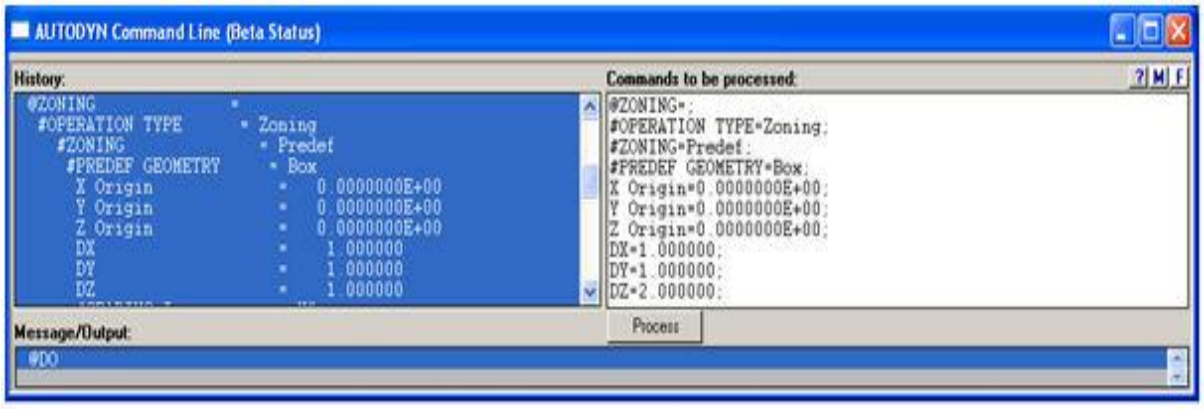
The easiest way to use the command line and become familiar with the command structure is to use the user interface to zone and fill parts as usual. The commands will then be recorded and can then be edited and played back as described above. Also, as can be seen in the picture above some guidance is given in the history window for the syntax used to set options and parameters etc. should you choose to type in the commands directly.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Undock Command Line



Press this button to detach the command line panel from the main user interface and open it in a separate dialog window. If this window is then closed, the Command Line docks back into the main user interface.



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

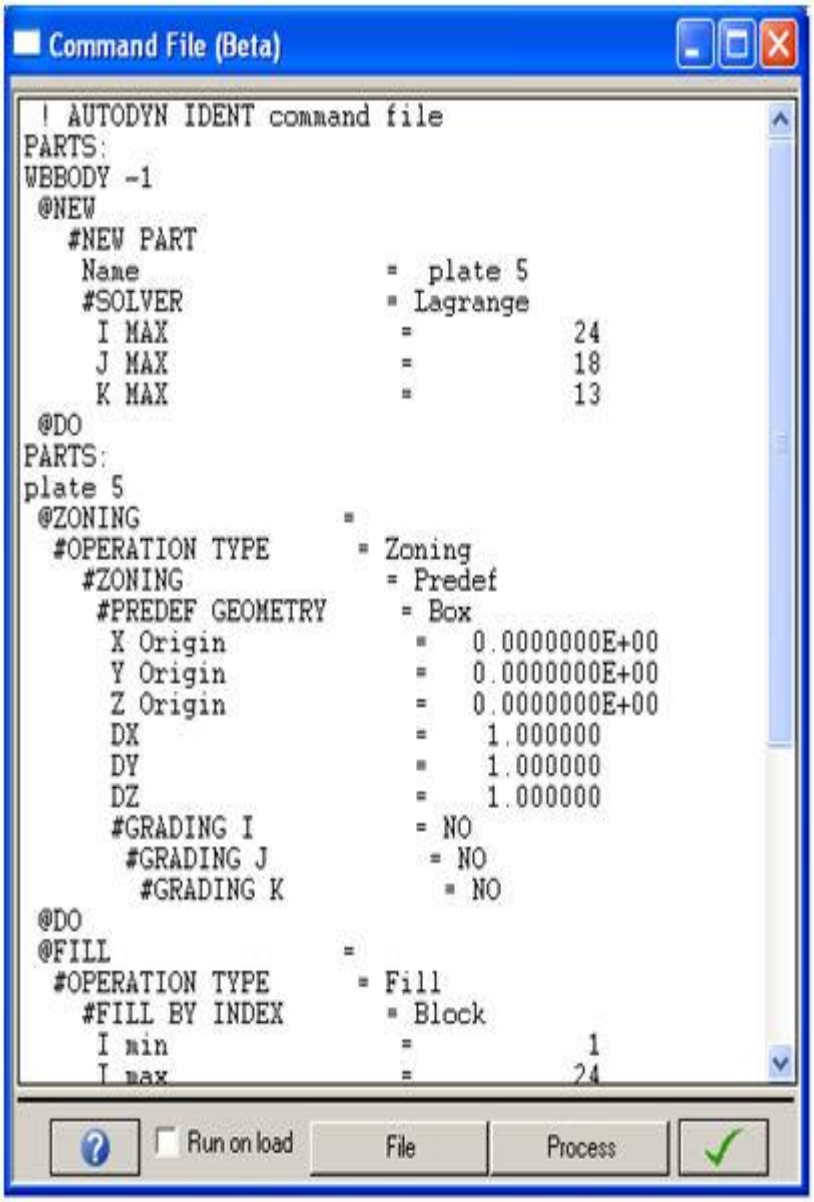
When the command line is detached from the main UI the command dialog panel contains three separate lists. The 'Command History' is shown on the left hand side and contains all valid commands executed in the current session. Multiple commands can be selected from this list to be replayed. The Message/Output panel is shown at the bottom. This provides the user with help if they request it and gives any error messages when invalid commands are performed.

The 'Commands to be processed:' list behaves the same as the command line when docked into the main UI but is much more user friendly and allows multiple lines to be input before pressing the process button. Any commands selected from the History list are copied into the 'commands to be processed:' panel and can then be edited before being processed.

The example above shows how you can replay a zoning operation. In this case the DZ parameter is modified before the command is re-processed.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Command File Window



The command file window allows you to open an existing AUTODYN command file using the 'File' button.

The commands can be executed/modified in a number of ways. Select all or a number of the commands and press the process button. Alternatively, if you want to modify the commands before execution, simply select the required commands. The commands are then copied to either the single command line or the 'Commands to be processed:' list (if the command line is not docked to the main UI). The commands can then be modified before execution by either pressing return on the single command line or the process button in the Command line panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

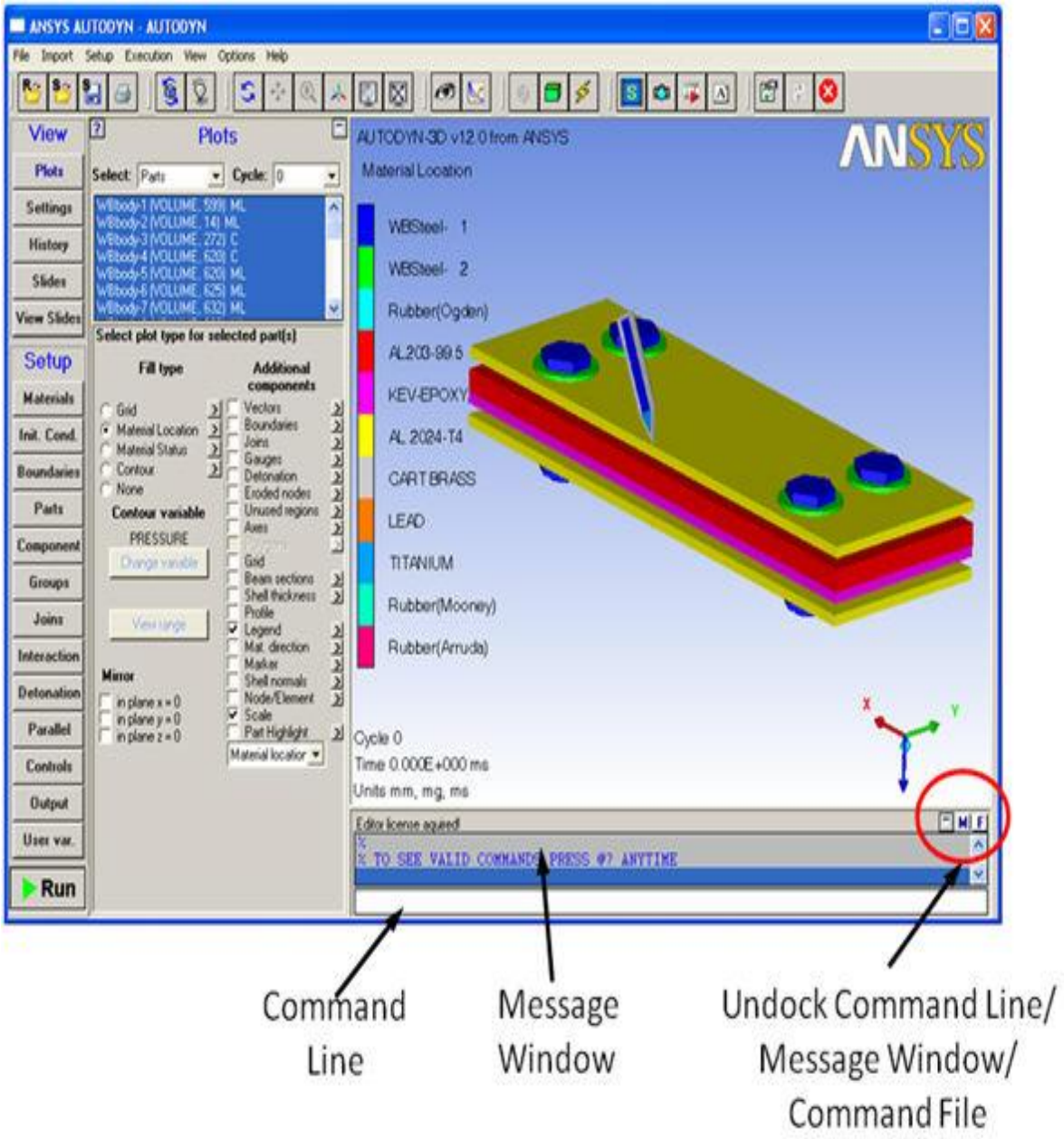
If the 'Run on Load' option is selected the commands in the selected file can automatically be run on loading the file, or alternatively the user is given the option to interactively step through the commands.

If stepping through the command file the user is shown the next command before processing it. If at any stage 'No' is selected AUTODYN stops reading the file.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Message Panel

The message panel shows a variety of messages during problem setup, execution and viewing.

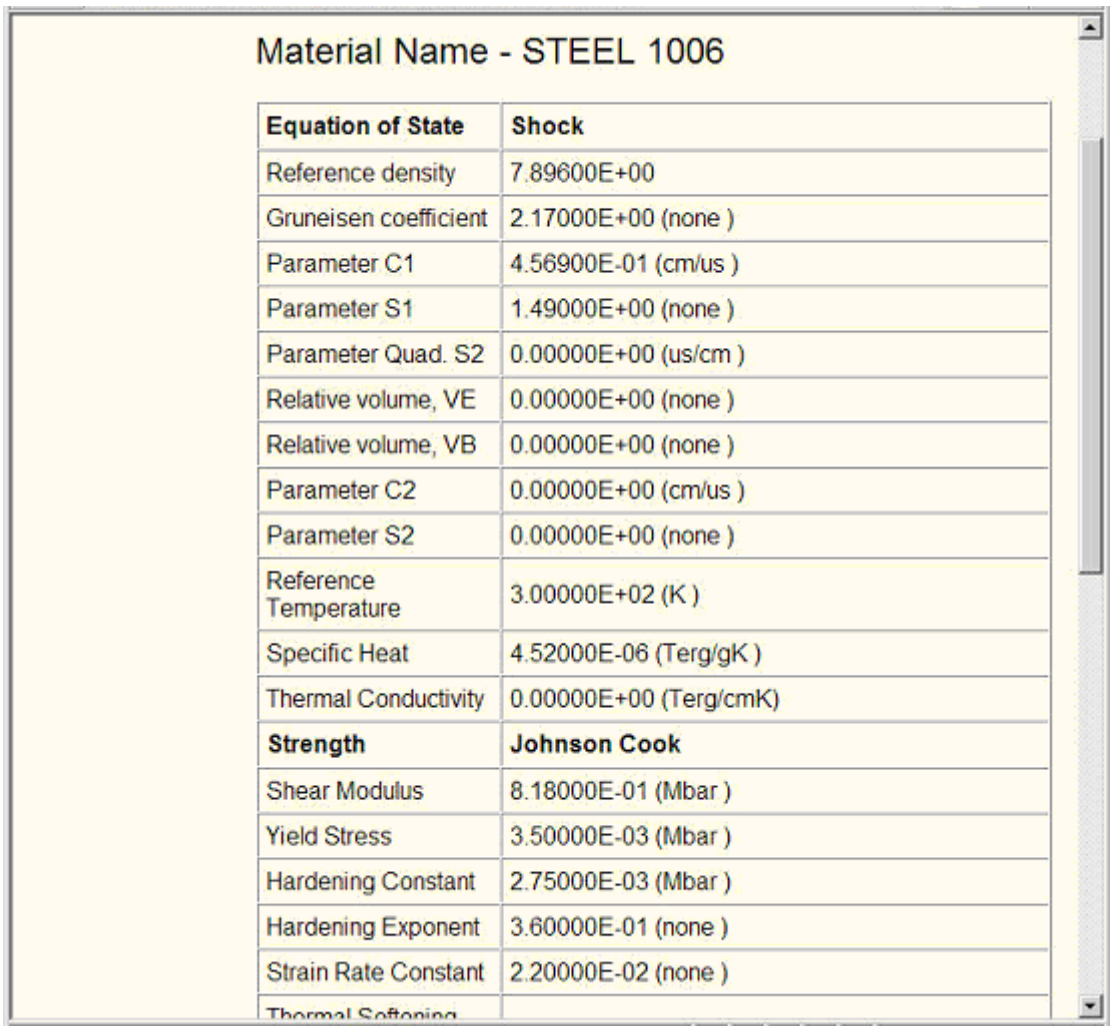


Clicking the **M** box at the top right corner of the message panel will open a scrollable window showing all the messages displayed during your session.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - HTML review windows

While working on a model, you may want a summary of particular data such as material properties, detonation point or gauge point locations. These are created as an HTML document and displayed in a browser window. You can easily insert these documents into reports and publications.



Material Name - STEEL 1006

Equation of State	**Shock**
Reference density	7.89600E+00
Gruneisen coefficient	2.17000E+00 (none)
Parameter C1	4.56900E-01 (cm/us)
Parameter S1	1.49000E+00 (none)
Parameter Quad. S2	0.00000E+00 (us/cm)
Relative volume, VE	0.00000E+00 (none)
Relative volume, VB	0.00000E+00 (none)
Parameter C2	0.00000E+00 (cm/us)
Parameter S2	0.00000E+00 (none)
Reference Temperature	3.00000E+02 (K)
Specific Heat	4.52000E-06 (Terg/gK)
Thermal Conductivity	0.00000E+00 (Terg/cmK)
Strength	**Johnson Cook**
Shear Modulus	8.18000E-01 (Mbar)
Yield Stress	3.50000E-03 (Mbar)
Hardening Constant	2.75000E-03 (Mbar)
Hardening Exponent	3.60000E-01 (none)
Strain Rate Constant	2.20000E-02 (none)
Thermal Softening	

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - File names

AUTODYN file names have the underlying format of ***Name_information.extension***.

Examples:

A cycle zero Save file would be named **myfilename_0.ad**

The Save file for cycle 12500 would be named **myfilename_12500.ad.**

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - On-Line help

The documentation and help pages for AUTODYN have been merged into a single context sensitive HTML documentation system.

All documents are accessible from the AUTODYN graphics user interface.

Pressing (Help) bring up context sensitive help pages in your default web browser.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Supported Operating Systems and Compilers

The following operating systems are supported at release 12.0 of ANSYS AUTODYN.

System	C++	Fortran
Windows XP Pro, 32 bit Windows Vista, 32 bit	Microsoft Visual Studio C++ .NET 2005	Intel Fortran 10.1
Windows XP Pro, 64bit Windows Vista, 64bit	Microsoft Visual Studio C++ .NET 2005	Intel Fortran 10.1
RedHat EL 4 (Update 5)		Intel Fortran 10.1
SUSE 10		Intel Fortran 10.1

32 Bit Linux

RedHat Enterprise Linux AS release 4 (Update 5)

64 Bit Linux (x86-64)

SUSE Professional 10

64 Bit Linux (Itanium II)

RedHat Enterprise Linux AS release 4 (Update 5)

Third Party Software Requirements for Parallel

System	Parallel Environment
Windows XP Pro, 32 bit Windows Vista, 32 bit	WMPI 1.6.1
Windows XP Pro, 64 bit Windows Vista, 64 bit	WMPI 2.4.2
RedHat EL 4 (Update 5)	HP-MPI*
SUSE 10	HP-MPI*

*See Chapter 1.1 (Prerequisites for Running Distributed ANSYS) of the "Configuring Distributed ANSYS" section of the Installation and Licensing Documentation for the version of HP-MPI that is supplied as part of the installation.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - Initialization Settings

AUTODYN is installed with a pre-configured file called **autodyn.ini**, which is in the users Application Data directory, in the sub folder Ansys\v120\AUTODYN. This is the contents of the file after installation :

AUTODYN_initialization_file
application_window_width
1000
application_window_height
700
message_window_visibility
0
manual_refresh_default
0
background_color
1
hardware_acceleration
1
reset_plot_flags
0
plot_wireframe
0
plot_perspective
0
model_interactivity_default
0
graded_background
1
OpenGL_for_slides
1
remove_joined_faces
0
open_new_browser_window
1
enable_beep
0
mouse_option_rotate
2
keyboard_option_rotate
0
mouse_option_zoom
3
keyboard_option_zoom
0
mouse_option_translate
2
keyboard_option_translate
2

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

mouse_option_toolbar
1
keyboard_option_toolbar
0
mouse_option_examine
1
keyboard_option_examine
3
mouse_option_boxzoom
3
keyboard_option_boxzoom
2
use_prepost_only
0
additional_user_title

You can alter the contents of this file to customize the startup of AUTODYN.

You can use any text editor to make your changes (e.g. Notepad or MS Word).

You can set the following parameters in this file.

Application window width

This sets the width of the AUTODYN application window at startup.

Application window height

This sets the height of the AUTODYN application window at startup.

Message window visibility

This switch sets the visibility of the message window on startup.

0 Message window hidden
1 Message window visible

Manual refresh default

This switch sets **manual refresh** on or off. For large models you may want to have full control on the refreshing of the plot window.

0 Manual refresh off
1 Manual refresh on

Background color

This switch sets the color of the background for the **View Panel** on startup.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

0 Black background
1 White background

Plot wireframe

This switch sets the wireframe plotting option on or off on startup.

0 Plot wireframe off
1 Plot wireframe on

Plot perspective

This switch sets the perspective plotting option on or off on startup.

0 Plot perspective off
1 Plot perspective on

Hardware acceleration

This switch sets **hardware graphics acceleration** on or off on startup.

0 Hardware graphics acceleration off
1 Hardware graphics acceleration on

Model Interactivity default

This switch sets the default model interaction setting for the toolbar mouse/keyboard combination on startup.

1 Translate model
2 Rotate model
3 Zoom in/out

Reset plot flags

This switch sets **reset plot flags** on or off.

If this flag is on, when you load a new model (or a different cycle of the current model) the plot settings for all Parts will be reset to default values (e.g. default view, material locations, etc.).

If this flag is off, when you load a new model (or a different cycle of the current model) with a model already present, the plot settings (grid, material location, material status, contour) for the current model are migrated to the new model for each Part with the same name.

0 Reset plot flags off
1 Reset plot flags on

Graded background

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This switch sets **graded background** on or off. If this switch is on a graded background color is used. If this switch is off a solid black or white color is used depending on **Background color** setting.

0 Graded background off
1 Graded background on

OpenGL for Slide creation

This switch sets **OpenGL for Slide creation** on or off. Using OpenGL during the creation of slides can cause problems for some graphics cards. If you experience any problems creating slides with this switch turned on, try turning it off.

0 OpenGL off for slide creation
1 OpenGL on for slide creation

Browser Options

By default, when AUTODYN opens an html document (either for the on-line help, or for the summary reviews), the page will be displayed in any browser that is currently open. By setting this flag to 1, a browser will be opened to display the information. This option may not work for all browsers.

Warning Sound Options

This option lets you enable or disable the warning beep

Mouse and Keyboard options

These options let you set the mouse and keyboard combinations for interaction. Each action type has a keyboard and an associated mouse option. A default will be used if any are not defined.

Mouse
0 Not used
1 Left Mouse Button
2 Middle Mouse Button
3 Right Mouse Button

Keyboard
0 None
1 Shift
2 Control
3 Alt

Mouse Option Rotate & Keyboard Option Rotate	Rotate mouse and keyboard combination
Mouse Option Zoon & Keyboard Option Zoom	Zoom mouse and keyboard combination

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Mouse Option Translate & Keyboard Option Translate	Translate mouse and keyboard combination
Mouse Option Toolbar & Keyboard Option Toolbar	Toolbar mouse and keyboard combination

This is the default combination for the interaction type selected on the toolbar

Mouse Option Examine & Keyboard Option Examine	Examine mouse and keyboard combination
Mouse Option Boxzoom and Keyboard Option Boxzoom	Box zoom mouse and keyboard combination

PrePost license usage

By default this switch is set to zero and AUTODYN will start up using a solver license. When the switch is set to zero AUTODYN will start up using a PrePost license only.

Additional user title

This option lets you add additional information below the top left hand corner of the **View Panel.**

You can enter up to 256 characters. The Additional User Title should always be the last option in the file.

Command Line Options

This feature has Beta status and can be activated by the user by adding the following lines above the Additional User Title option in the autodyn.ini file:

command_line_on
1

By activating this flag you accept that this is a Beta option and therefore may not be as stable or robust as expected in a general release version.

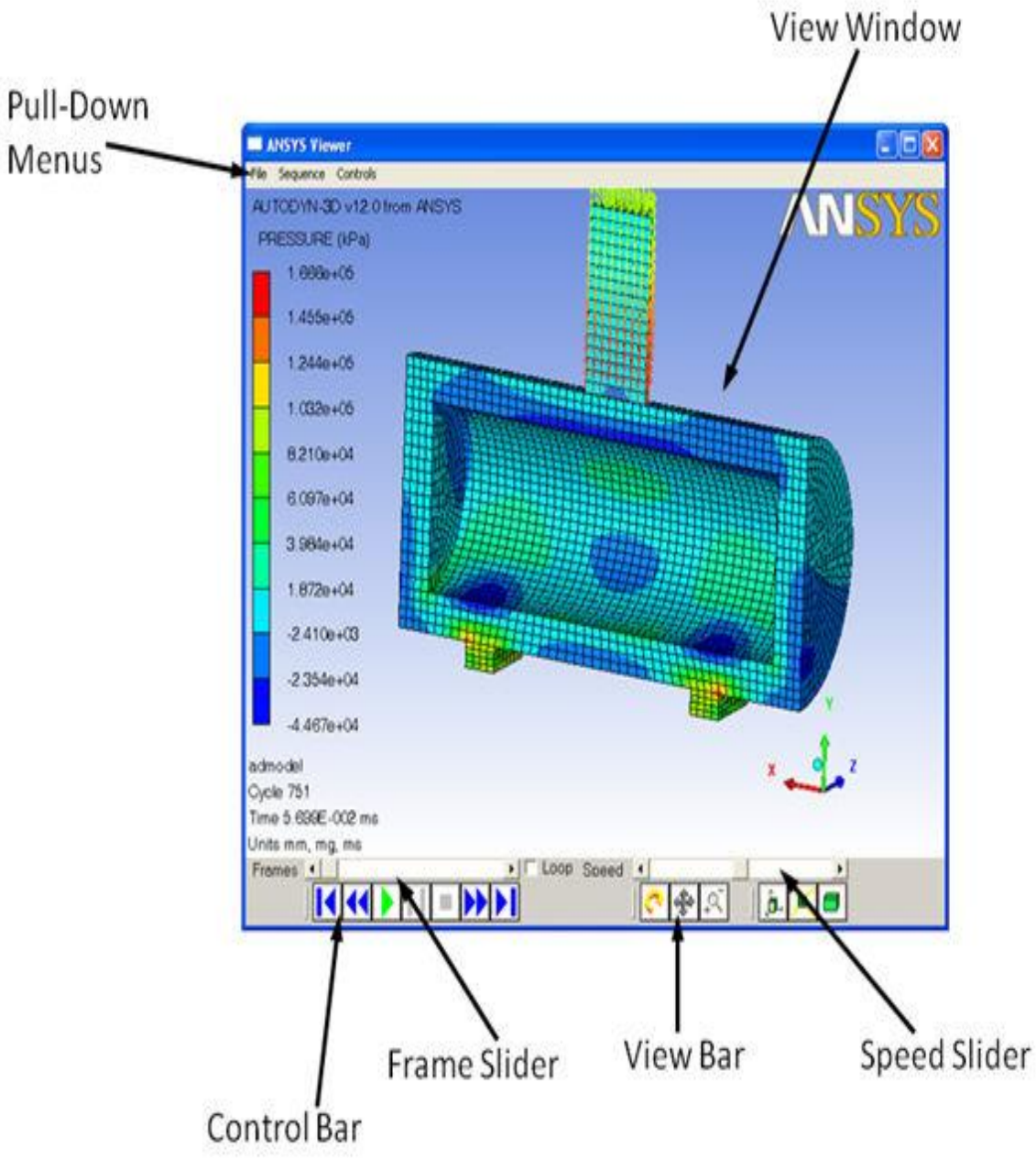
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Slide Viewer

The View Slides option on the Navigation Bar starts the Slide Viewer (it can also be run independently of AUTODYN).

The Viewer lets you load and view GIF and GFA animations.

Components of the Viewer's user interface :



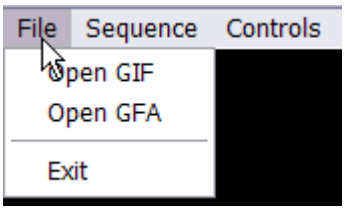
View Window

This is where your animations are shown.

Pull-down Menus

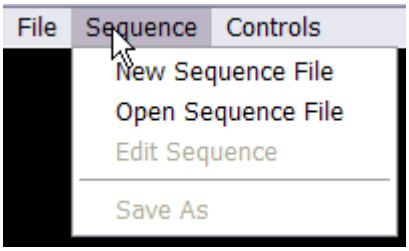
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

File



This pull-down menu lets you load single GIF and GFA animation files for playback, and exit the program.

Sequence



This pull-down menu lets you playback multiple GIF animations using sequence files.

A sequence file contains a list of GIF animations you want to view consecutively.

All the animations in a sequence file must be in the same directory as the sequence file.

New Sequence File

This option opens a new sequence file.

Open Sequence File

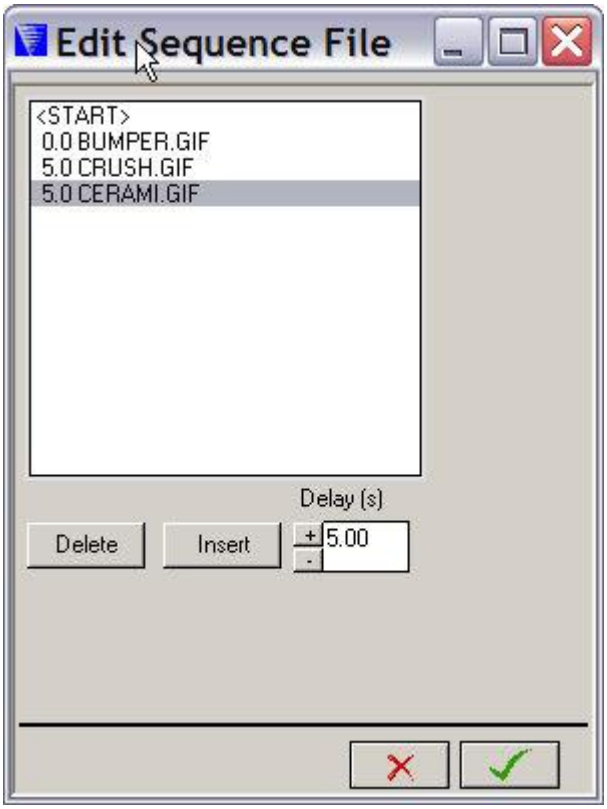
This option opens an existing sequence file.

Edit Sequence

This option lets you insert and delete animations in the current sequence file.

The following dialog window appears when you select this option :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Sequence list

The box at the top of this dialog contains a list of the animations currently included in the sequence, together with their time delay (how long the first slide in the sequence is shown, in seconds).

Insert

Press this button to insert another animation immediately after the selected animation.

Delete

Press this button to delete the selected animation.

Delay

Enter a time delay in this field (in seconds). When you **Insert** an animation, this time delay is applied.

You can create and edit sequence files manually if you wish. Sequence files must have a **.seq** extension and use the following format

Format	**Example**
The number of animations in the sequence	3

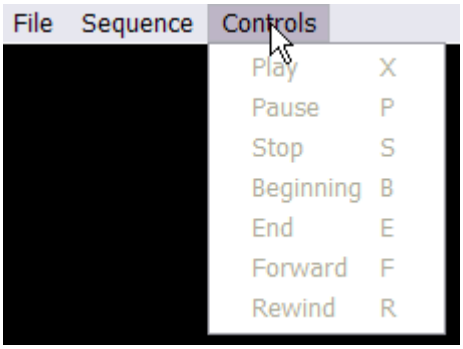
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

	0.0 anim.gif
Default frame delay in seconds + <space> + file name	1.2 another.gif
Default frame delay in seconds + <space> + file name	1.6 yetanother.gif
Default frame delay in seconds + <space> + file name	
etc.	

Save As

This option saves the current sequence file (**.seq**)

Control



This pull-down menu provides the same functions that are available in the Control Bar.

Control Bar

This bar lets you control the playing of the GIF or GFA animation you have loaded.

Pressing a button on this bar performs the following actions (you can type the corresponding **Hotkey** as an alternative to pressing the a button).

Button	Hotkey	Action
	b	Go to the first frame of the animation
	r	Go back one frame
	x	Play (from the current frame)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

p	Pause (at the current frame). This is a toggle button.
s	Stop (at the current frame)
a	Advance one frame
e	Go to the last frame of the animation

View Bar

This bar only appears when you have loaded a GFA animation.

The buttons on this bar let you control how you view your GFA animation and work in the same way as the corresponding icons in the AUTODYN **Tool Bar**

Rotate view
Translate view
Zoom in or out
Set view (brings up a window that allows you to set an exact view)
Reset view (to the initial view)
Turn perspective on or off (toggle)

Frames Slider

This slider lets you move quickly to any frame in the animation(s).

Speed Slider

This slider lets you adjust the playback speed for animations.

Loop

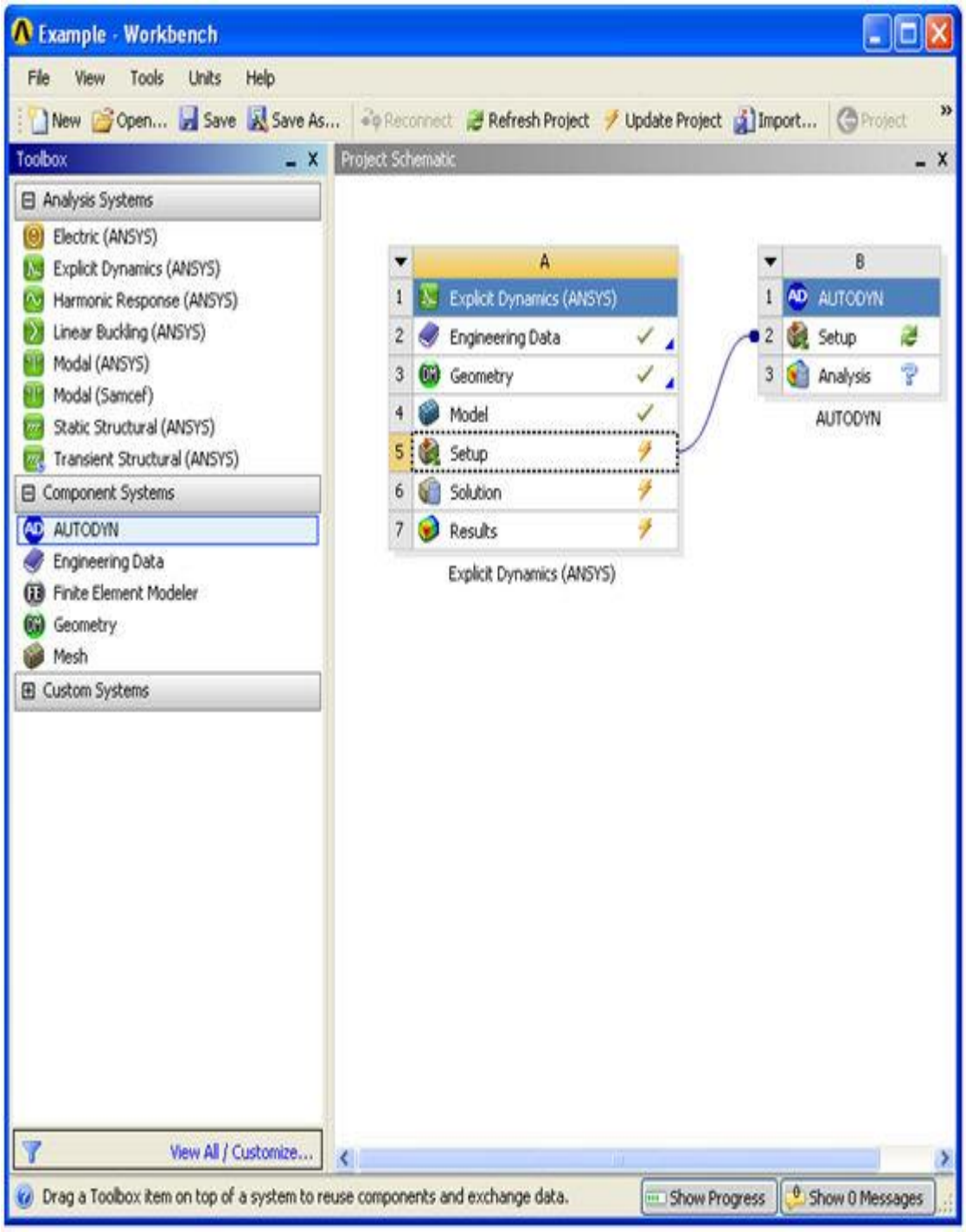
Check this box if you want the animation(s) to automatically start again from the first frame when the last frame is reached.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - AUTODYN in Workbench

An overview of how to access AUTODYN in Workbench at release 12 is provided below. Full guidance on how to use Workbench at release 12 is provided in the on-line help system of Workbench.

A new paradigm for managing simulations is used with the ANSYS Workbench GUI at release 12. The ANSYS Workbench GUI now consists primarily of a Toolbox region, Project Schematic, Toolbar, and menu bar. Depending on the analysis type and/or application or workspace, you may also see other windows, tables, charts, etc



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

One way to work in ANSYS Workbench is to drag an item such as a component or analysis system from the Toolbox to the Project Schematic or to double-click on an item to initiate the default action. You can also use the context menus, accessible from a right-mouse click, for additional options. You will view your analysis systems -- the components that make up your analysis -- in the Project Schematic including all connections and links between the systems. The individual applications in which you work will display separately from the ANSYS Workbench GUI, but the results of the actions you take in the applications may be reflected in the Project Schematic.

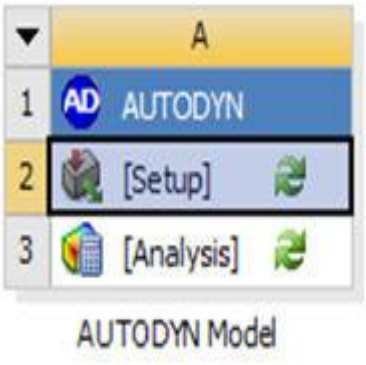
Full details of how to use ANSYS Workbench are provided in *"Workbench Help"* section of the Workbench on-line help system.

An ANSYS AUTODYN product license should enable three primary systems in the Toolbox of the Workbench GUI.

AUTODYN System

The AUTODYN system is located in the Component Systems toolbox of the Workbench GUI. This system allows you to start-up the traditional ANSYS AUTODYN application and includes pre-processing, solve and post-processing options for all AUTODYN solvers.

To start-up the AUTODYN application simply insert an AUTODYN system into the project schematic. Then double click the Setup cell.



The AUTODYN application will start-up and you can setup, solve, post-process models in the familiar AUTODYN user interface. The default start up mode will be the generation of a new model. The start up mode can be changed by right-mouse click the Setup cell after which you can select:

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

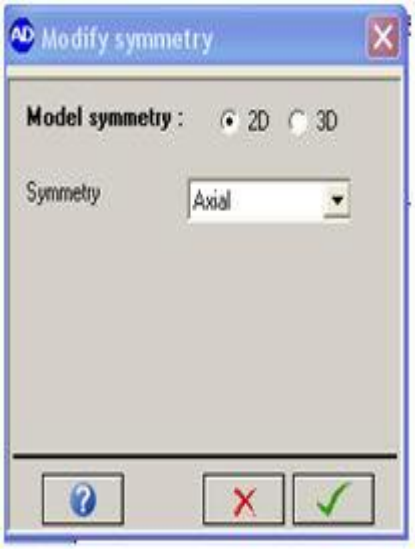
- New Model - default
- Import Model - open existing AUTODYN Save file
- Select user executable - defines where a user customized version of AUTODYN resides
- Duplicate - makes a copy of the model file
- Transfer Data From - Model data from either the Mesh system or the Explicit Dynamics system are imported into AUTODYN

After selection of the desired Setup cell setting double clicking the Setup cell again will start up AUTODYN in the required mode.

Notes:

- Model Name: The database for an AUTODYN model will now always be admodel_0.ad. The project name is the now unique identifier for all databases (see below).
- 2D/3D symmetry: By default, the AUTODYN application assumes you are performing a 3D simulation in the default unit set (mm, mg, ms). To define 2D axial or planar symmetry, use the *Setup, Symmetry* option in the toolbar after start-up of the AUTODYN application.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



- Saving a project: To save a model, the recommended approach is to save the Workbench project either from within the AUTODYN application (File, Save Project) or from the project page of Workbench. This ensures all data used in the AUTODYN Simulation is saved to disk.
- Exporting an AUTODYN database: The AUTODYN database (restart file) can be exported to a file (independent of the Workbench project) using the *File, Export* options in the AUTODYN application.

Please refer to the "*Component Systems*, AUTODYN" section of the Workbench On-line help for further details on the operations and properties available for an AUTODYN system.

Explicit Dynamics (ANSYS) system

The Explicit Dynamics (ANSYS) system is located in the "Analysis Systems" toolbox of the Workbench GUI. This system allows you to completely setup, solve and post-process an explicit dynamics simulation with the ANSYS Mechanical Workbench user environment. At release 12, this user interface fully supports the 3D Finite Element features of ANSYS AUTODYN (unstructured Lagrange solvers, isotropic inert materials and linear orthotropic materials).

The system contains a number of unique features for explicit dynamic simulation inside Workbench:

- Unidirectional and bidirectional links to CAD geometry
- Links to Design Modeler

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

- Advanced meshing capabilities tailored for Explicit analysis
- Mesh independent loading and boundary conditions
- Fully parameterized engineering data, geometry, initial conditions, loads, boundary conditions
- Automated "What If" analysis capability
- Links to design exploration tools

The full capabilities of the ANSYS Explicit Dynamics system are described in the online help for the Mechanical analysis systems in Workbench.

Models setup in the ANSYS Explicit Dynamics system can be persistently linked to an AUTODYN system to transfer material data, mesh, connections, loads and constraints, analysis settings etc... into ANSYS AUTODYN.

For modeling complex geometries in ANSYS AUTODYN, we recommend you define the FE components of the model in an ANSYS Explicit Dynamics system then transfer the model setup to the AUTODYN system and add on your Euler-SPH regions and explicit FSI/Coupling as required.

Mesh system

The Mesh system is located in the Component Systems toolbox of the Workbench GUI. This system allows you to link to your model geometry and generate a mesh. The mesh can then be transferred to an AUTODYN system for subsequent analysis.

This system allows you to use a similar workflow that was available at release 11 to link AUTODYN models to CAD/geometry and meshing tools. However with Version 12 the link is not persistent anymore, which means that with every update of the workflow system the definitions made in AUTODYN (material filling, boundary condition assignment, etc.) are lost and have to be re-applied.

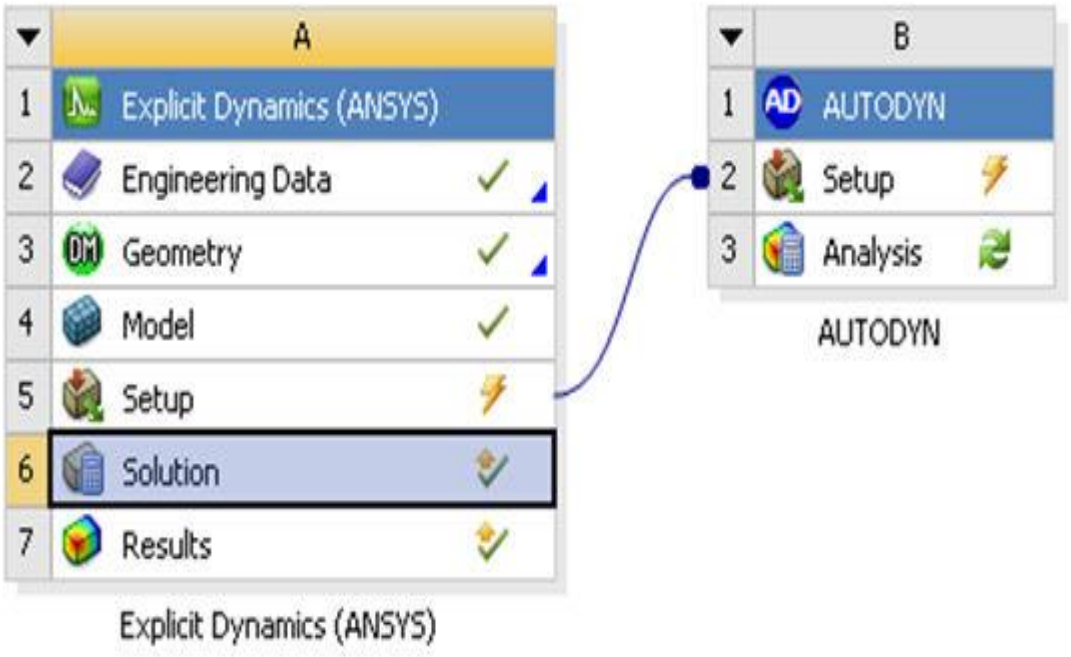
Therefore at release 12, we recommend using the ANSYS Explicit Dynamics system to link to CAD/geometry and generate a mesh for use in AUTODYN where loads, constraints and material assignments are maintained after any change to the geometry or mesh.

Linking different systems

If the mesh/geometry of the application to be solved in AUTODYN cannot be made with the simple model building tools available in the AUTODYN application, we recommend the following approaches to build the AUTODYN model.

ANSYS Explicit Dynamics => AUTODYN

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



The recommended approach for bringing Lagrangian FE based models with complex geometries into AUTODYN is to start off with an ANSYS Explicit Dynamics system.

- Define material data using the Engineering Data application
- Link directly to CAD Geometry or use DesignModeler to generate geometry directly or adapt CAD geometry for simulation purposes
- Mesh the geometry and define connections (Contact, Joins, Spotwelds)

Optionally:

- Assign initial velocities to components
- Apply loads and constraints directly to the geometry
- Define solution controls and outputs
- Solve and Post process results

To transfer the model from an ANSYS Explicit Dynamics system to the AUTODYN application, link the Setup cell of the ANSYS Explicit Dynamics system to the Setup cell of the AUTODYN system. Open the AUTODYN application by double click on the AUTODYN system Setup cell. All relevant model information defined in Engineering Data, Geometry, Model and Setup components of an ANSYS Explicit Dynamics system are transferred into the AUTODYN application.

Once in the AUTODYN application, you can complete the model setup (for example, add Euler and SPH regions) then solve in the AUTODYN application.

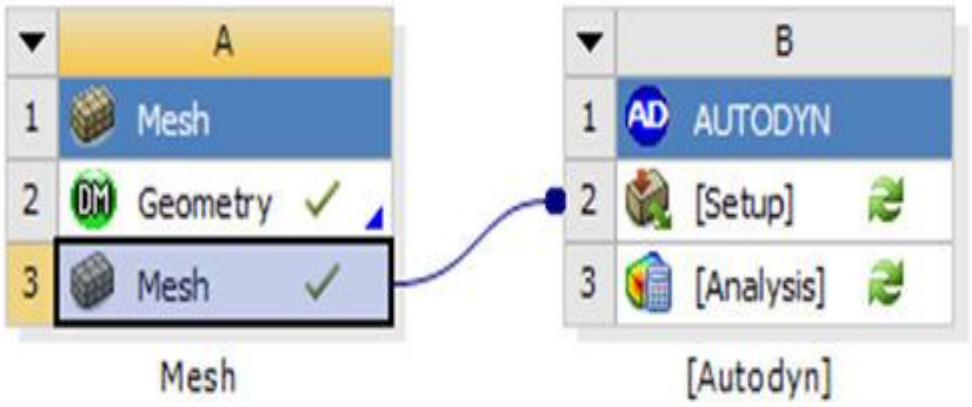
The link generated between the AUTODYN and ANSYS Explicit Dynamics system is persistent. You can make changes to the model defined in the ANSYS Explicit Dynamics system, update the ANSYS Explicit Dynamics system Setup cell, then update the AUTODYN system Setup cell. The latest model definition will be transferred into the AUTODYN model while retaining additional features present only in the AUTODYN application. This makes it very easy to make changes to model.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Notes:

- Changes made to the model in the AUTODYN application are NOT transferred back to the ANSYS Explicit Dynamics system. The link is one way.
- To avoid overwriting changes you make in the AUTODYN application, we recommend that if a portion of the model is defined in the ANSYS Explicit Dynamics system (for example and initial velocity) then only make changes to this in the ANSYS Explicit Dynamics system. Transfer to the AUTODYN model using the update procedure described above.
- Local co-ordinate systems defined and used in the ANSYS Explicit Dynamics system are transferred through to the AUTODYN application however they cannot be reviewed or modified in AUTODYN.
- Displacement boundary conditions defined in the ANSYS Explicit Dynamics system are converted to equivalent velocity boundary conditions in the AUTODYN application.
- Loading defined in tabular form in the ANSYS Explicit Dynamics system cannot be edited or reviewed in the AUTODYN application. The data is transferred and used in the AUTODYN simulation.
- Spot weld breaking criteria defined in the ANSYS Explicit Dynamics system will be converted from a stress to a force criterion

Meshing => AUTODYN



If you only want to transfer a Mesh (of complex geometry) into AUTODYN then this can be achieved (as in release 11.0) using a Mesh system. Start by inserting a Mesh system from the Components toolbox.

- Link directly to CAD Geometry or use DesignModeler to generate geometry directly or adapt CAD geometry for simulation purposes
- Mesh the geometry
- Create Named Selections

To transfer a mesh from the Mesh to the AUTODYN application, link the Mesh cell of the Mesh system to the Setup cell of the AUTODYN system. Open the AUTODYN

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

application by double clicking on the AUTODYN system Setup cell. The mesh and Named selections will be transferred to AUTODYN.

With Version 12 the link generated between the AUTODYN and Mesh system is not persistent anymore. This means that with every update of the workflow system the definitions made in AUTODYN (material filling, boundary condition assignment, etc) are lost and have to be re-applied.

Therefore at release 12, we recommend using the ANSYS Explicit Dynamics system to link to CAD/geometry and generate a mesh for use in AUTODYN, where loads, constraints and material assignments are maintained after any change to the geometry or mesh.

Notes:

- Connections defined in the Mesh system will not be transferred to the AUTODYN application.
- Material assignments, Boundary Conditions and Loads defined in the AUTODYN application will be lost during the update and will have to be re-applied.

FEModeler

Legacy and third party FE model data can be imported into the FEModeler system in Component toolbox. This data can be transferred into AUTODYN via a Mesh system or an ANSYS Explicit Dynamics system.

Note: Direct transfer of data from the FEModeler application into AUTODYN is not possible at release 12.

Data Transferred by linking different systems to AUTODYN

Node Coordinates

The nodal coordinates are transferred.

Mesh Elements

The mesh elements are transferred, noting that Pyramid elements are not supported within AUTODYN, so these are converted into two tetrahedra.

Body Definitions

The definitions of bodies in the Workbench are imported as Parts in AUTODYN.

In addition, for Meshing, Simulation and Finite Element Model Objects, the following are transferred.

Parts Definitions

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The definitions of parts (both multi body parts and single body parts) are shown as Components in AUTODYN.

Named Selection

The named selections are represented as Groups in AUTODYN. Note that the face named selection is transferred as a face group; all other named selections are transferred as node groups.

Unit Setup

All models are transferred to AUTODYN in units of mm, mg, ms.

Material Associations

Each Body (AUTODYN Part) transferred is assigned a separate material.

Shell thickness

Shell parts will be assigned with the thickness as defined in the Workbench object. An initial condition will be created for each part.

Beam Cross-Sections and Orientations

The beam cross sections associated to elements and their orientations are transferred and applied correspondingly.

In order to update the model in AUTODYN to that in the currently selected project page item, use the 'Update Model from Workbench' option, found in the Import menu or on the toolbar. This function will update the parts that are in AUTODYN to reflect the mesh in Workbench. If a part contains a single material then the updated parts will be refilled with any materials and initial conditions that are associated. New parts are added to the model as appropriate, deleted parts should be manually removed from the AUTODYN model.

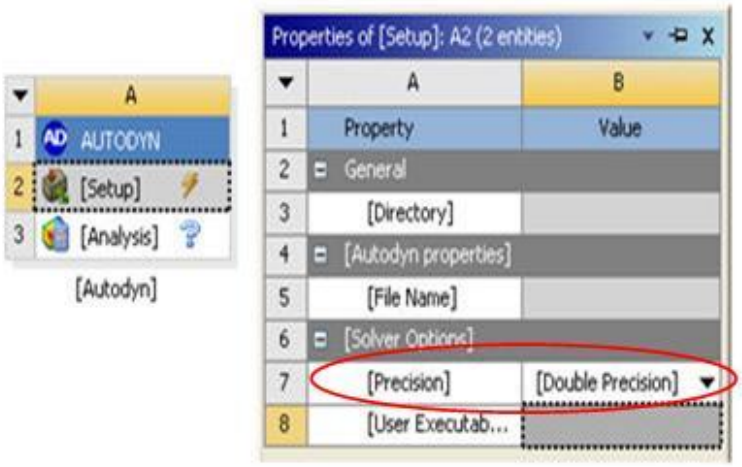
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Overview - AUTODYN solver options in Workbench

There are a number of AUTODYN solver options, which may be changed by right-mouse click the Setup cell of the AUTODYN system.

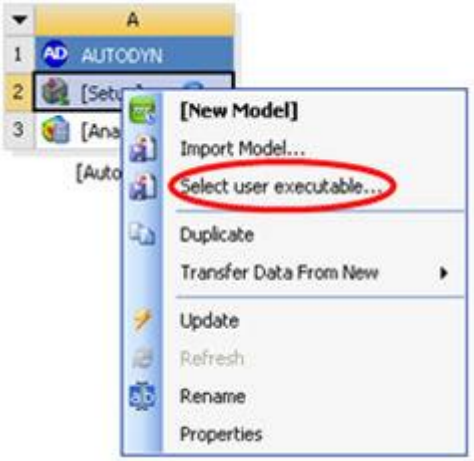
Precision

Note that by default, the single precision version of AUTODYN will be used. To run in double precision, from the ANSYS Workbench project schematic, right click the Setup cell of the AUTODYN system and select Properties after which the Properties window will pop up where you can select what type of executable you want to run.



User Executable Location

By default, Workbench will start the standard AUTODYN executable. You may also choose an executable created with a user-subroutine. To run your own user executable from the project schematic, right click the Setup cell of the AUTODYN system and left click Select user executable, after which you will be asked for the user executable name and locaton.



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menus

Pull-down Menus

Pull-down menus let you access all the features and dialog panels/windows in AUTODYN.

Most options can also be accessed through buttons on the Tool Bar and Navigation Bar.

Options that are only accessible through pull-down menus are shown below in red.

Options that only are available when AUTODYN is started outside the Workbench environment are shown below in green.

File	Import	Setup	Execution	View	Options	Help
New	from TrueGrid (.zon)	Description	Run Interactively	Plots	Interactivity	Contents
Open	from ICEMCFD (.geo)	Symmetry	Batch Runner	Plot Settings	Clipping Planes	About
Open Results File	from LS-DYNA (.k)	Material		History Graphs	Default Settings	
Save	from MSC.NASTRAN bdr (.dat)	Initial Conditions		Slideshow	Pause License	
Save Project	Convert IJK to unstructured	Boundary		Performance Profiler	Create User-Subroutine Project	
Save As		Parts				
Save As Version		Components				
Export to Version		Groups				
Manage Folders		Joins				
Save settings		Interactions				
Load settings		Detonation				
Print		Parallel				
Export PostScript		Controls				
Export VRML		Output				
Save Formatted		User Variables				
Export Formatted						
Open Formatted						

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Import Formatted						

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

File

Pull-Down Menu-File

This pull-down menu lets you perform file operations.

New

Lets you create a new model

Open

Lets you open (load) an existing model saved in a binary file.

Open Results Files

Lets you open an existing results file. For more details of results files, see the section Outputs - Results Files.

Save

Lets you save the current model using the current Ident for the filename to a binary file.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Save Project

Lets you save the Workbench project and ensures that the AUTODYN analysis model and all Workbench related data is saved to disk.

Save As

Lets you save the current model using a new Ident for the filename.

Save As Version

Lets you save the current model in a format supported by a previous version.

Export to Version

Lets you save the current model outside the Workbench project in a format supported by a previous version.

Manage Folders

Lets you manage your project folders.

Save settings

Lets you save the settings for the current plot in a settings file.

Load settings

Lets you load and apply plot settings from a saved settings file.

Print

Lets you output data for the current plot to a printer.

Export PostScript

Lets you output data for the current plot to a PostScript file.

Export VRML

Lets you output data for the current plot to a VRML file.

Save Formatted

Lets you save the current model using the current Ident for the filename to a formatted (ascii) file.

Export Formatted

Lets you save the current model to a formatted (ascii) file.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Open Formatted

Lets you open (load) an existing model saved in a formatted (ascii) file.

Import Formatted

Lets you open (load) an existing model saved in a formatted (ascii) file.

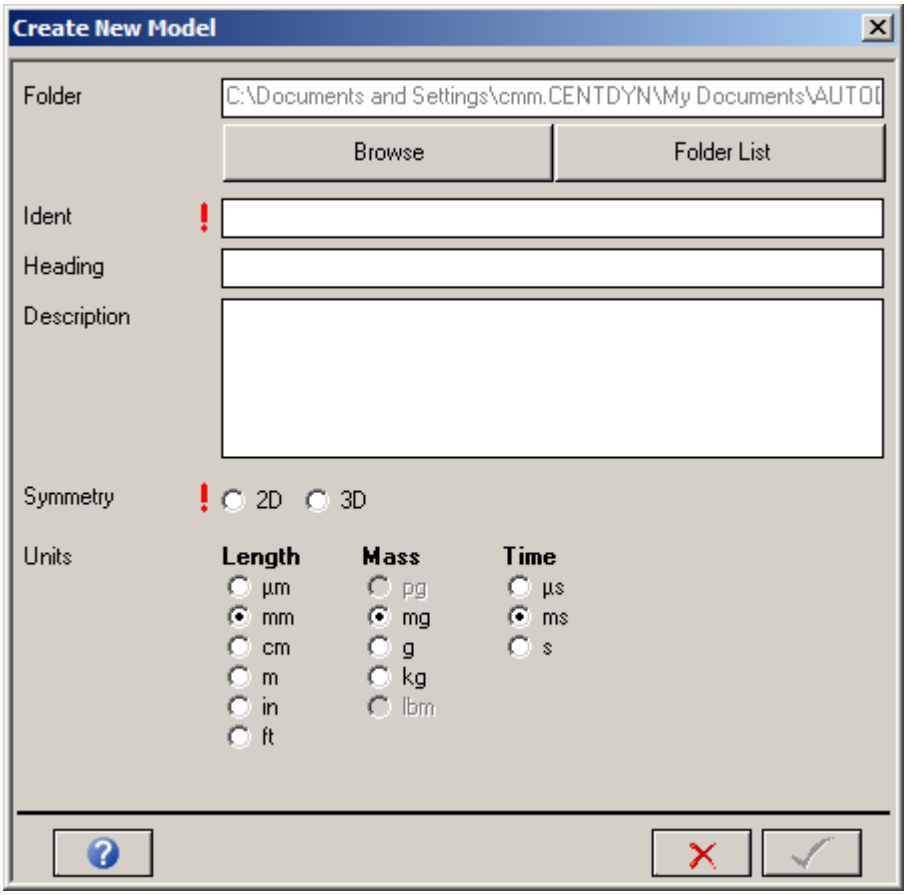
Close AUTODYN

Lets you exit AUTODYN.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - File - New

This menu is only available when AUTODYN is started outside Workbench.



This window lets you start the creation of a new model.

Folder

This box shows you the folder where files for the model will be kept.

Press the **Browse** button to select a different folder.

Press the **Folder List** button to select a folder from your current list of folders or manage your folder list

Ident

Enter an Ident for your model in this field.

The ident you choose is used to identify all the files created for your model.

e.g. If your Ident is **MyImpactProblem**, then the cycles zero Save file for the problem will be named **MyImpactProblem_0.ad.**

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Heading

Enter a heading for your model in this field.

Headings appear in plots and other output for your model.

Description

Enter a more detailed description of your model here if you want.

Symmetry

Select the dimension (2D or 3D) and symmetry you want to use for your model.

In 2D you must choose either **Axial** or **Planar** symmetry

In 3D you do not need to choose a symmetry, but you can optionally select **X**, **Y** and/or **Z** symmetry.

(e.g. if you select X symmetry, then the plane X=0 is assumed to be a symmetry plane).

Units

Select the units of length, mass and time you want to use for your model.

The default units (mm,mg,ms) will give good results, with small rounding errors, for most models.

US Customary Units

US customary units have been added to units options when creating a new model.

These are
- Length unit (inch or foot)
- Mass unit (pound)

These units can be written in short as "in", "ft", and "lb". One of the three time units can be used in conjunction with the US customary units: microsecond, millisecond, or second.

Note they cannot be used together with Metric Units. For example, you cannot use inch for length unit and kilogram for the mass unit.

The following table shows the unit for each type of physical variables in the US customary system:

	in, lb, s	**in, lb, ms**	**in, lb, μs**	**ft, lb, s**	**ft, lb, ms**	**ft, lb, μs**
Length	in	in	in	ft	ft	ft
Mass	lb	lb	lb	lb	lb	lb

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Time	s	ms	μs	s	ms	μs
Temperature	K	K	K	K	K	K
Density	$lb.s^2/in^4$	$lb.ms^2/in^4$	$lb.\mu s^2/in^4$	$lb.s^2/ft^4$	$lb.ms^2/ft^4$	$lb.\mu s^2/ft^4$
Inertia	$lb.s^2.in$	$lb.ms^2.in$	$lb.\mu s^2.in$	$lb.s^2.ft$	$lb.ms^2.ft$	$lb.\mu s^2.ft$
Volume	in^3	in^3	in^3	ft^3	ft^3	ft^3
Force	lb	lb	lb	lb	lb	lb
Pressure	psi	psi	psi	psf	psf	psf
Impulse	lb.s	lb.ms	lb.μs	lb.s	lb.ms	lb.μs
Velocity	in/s	in/ms	in/μs	ft/s	ft/ms	ft/μs
Work	lb.in	lb.in	lb.in	lb.ft	lb.ft	lb.ft
Strain Rate	1/s	1/ms	1/μs	1/s	1/ms	1/μs
Energy	lb.in	lb.in	lb.in	lb.ft	lb.ft	lb.ft
Energy/Unit Mass	in^2/s^2	in^2/ms^2	$in^2/\mu s^2$	ft^2/s^2	ft^2/ms^2	$ft^2/\mu s^2$
Energy/Unit Volume	lb/in^2	lb/in^2	lb/in^2	lb/ft^2	lb/ft^2	lb/ft^2
Fracture Energy	lb/in	lb/in	lb/in	lb/ft	lb/ft	lb/ft
Fracture Toughness Squared	lb^2/in^2	lb^2/in^2	lb^2/in^2	lb^2/ft^2	lb^2/ft^2	lb^2/ft^2
Specific Head Capacity	$in^2/s^2.K$	$in^2/ms^2.K$	$in^2/\mu s^2.K$	$ft^2/s^2.K$	$ft^2/ms^2.K$	$ft^2/\mu s^2.K$
Thermal Conductivity	lb.s/K	lb.ms/K	lb.μs/K	lb.s/K	lb.ms/K	lb.μs/K

Micron Unit System

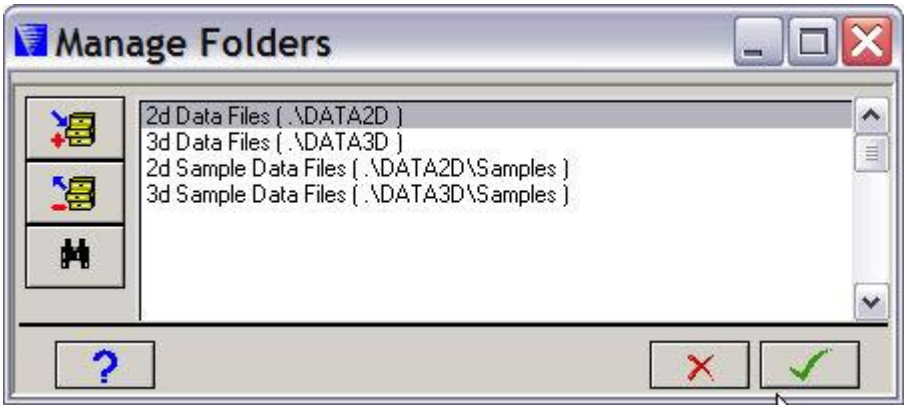
A new micron (μm) length system has been introduced. This can be used in conjunction with a mass unit of picogram (pg) and time unit of microsecond or millisecond.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Manage Folders

Pull-down Menu - File - Manage Folders

This menu is only available when AUTODYN is started outside Workbench


Manage Folders
2d Data Files (.\DATA2D)
3d Data Files (.\DATA3D)
2d Sample Data Files (.\DATA2D\Samples)
3d Sample Data Files (.\DATA3D\Samples)

This window lets you manage a list of the folders containing your AUTODYN models.

You can add and delete folders from this list as your projects change.

Folder List

The scroll box contains a list of the current folders you have selected. For each folder you see its alias (if defined) followed by its full path name in parentheses.

Double clicking on any folder in this list selects it as the directory for your current model.

Select / Create Folder

Press this button to add another folder to your list (using a browser). After selecting the directory for the folder, you can optionally specify an **alias** for the folder.

Remove Folder from list

Press this button to remove the selected folder from your list.

View Folder Contents

Press this button to view the model files in the selected folder.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Manage Folders - New Folder

This window lets you define an optional alias name for a folder.

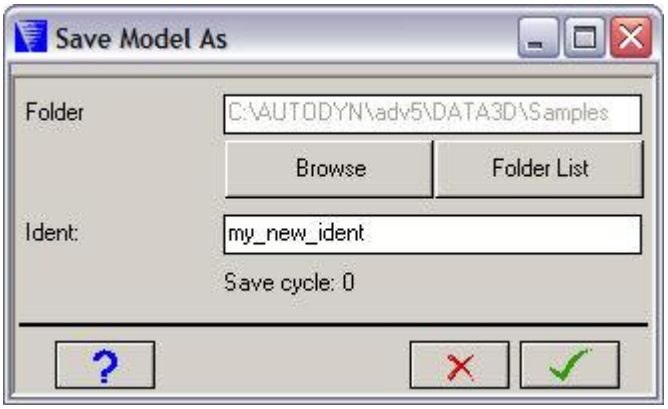
Enter the alias name in the field provided.

If you leave this field blank, the full folder name will be used to reference the folder.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - File - Save As

This menu is only available when AUTODYN is started outside Workbench



This window lets you save the current model under a new Ident.

Folder

This field shows the current folder in which model files are saved.

Browse

Press this button to browse to a new folder for saving model files.

Folder List

Press this button to select a folder from your folder list for saving model files and/or manage your folder list.

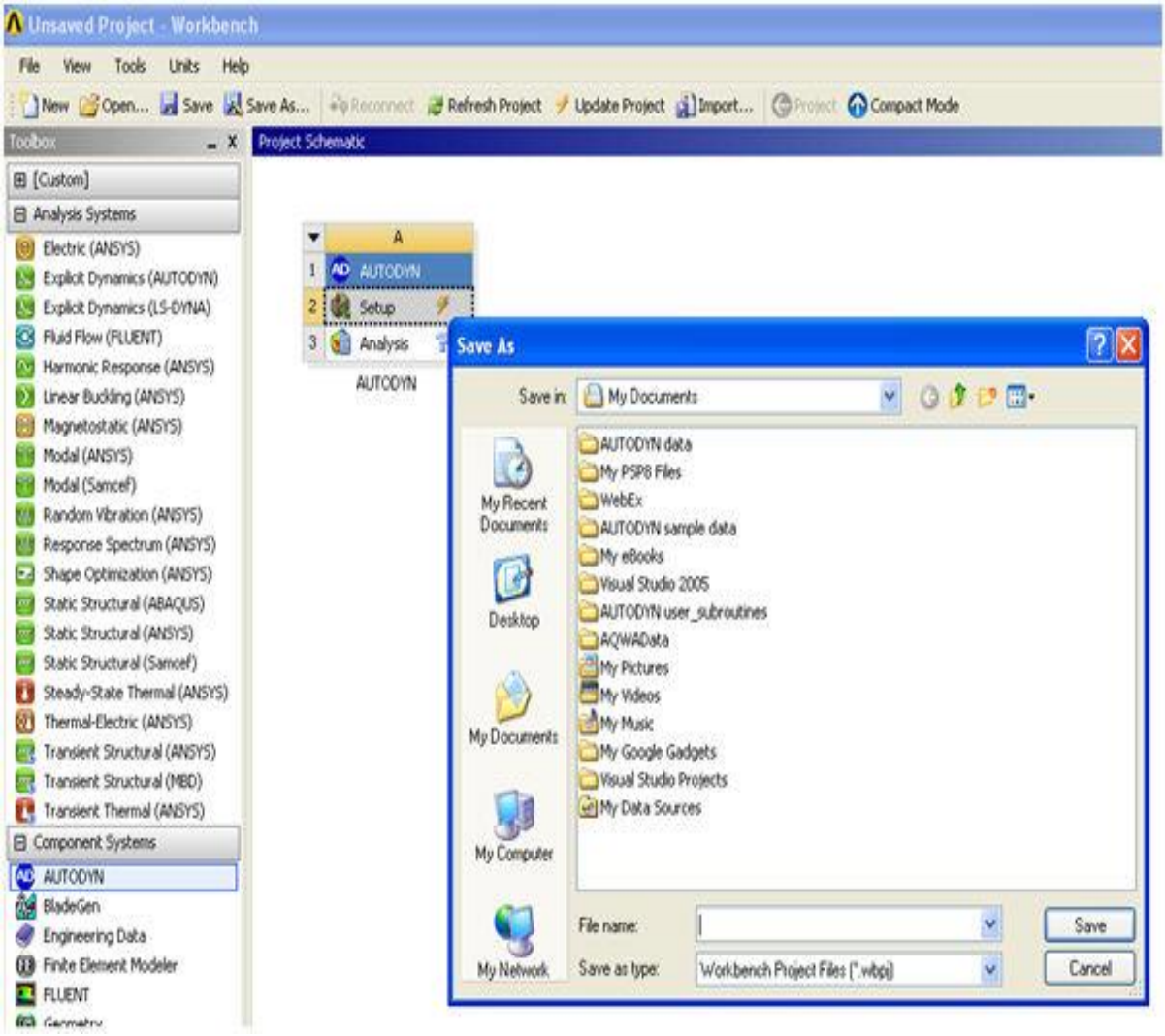
Ident

Enter the new Ident you want to use for saving model data.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

PullDown Menus - File - SaveProject

This option brings up the Workbench project page and allows you to save the current Workbench project and ensures that the AUTODYN analysis model and all Workbench related data is saved to disk.



Save in

This field shows the current folder in which the Workbench project tree is saved and the user can select a different folder if required.

File Name

The user defined project name.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - File - Save As Version

This menu is only available when AUTODYN is started outside Workbench.

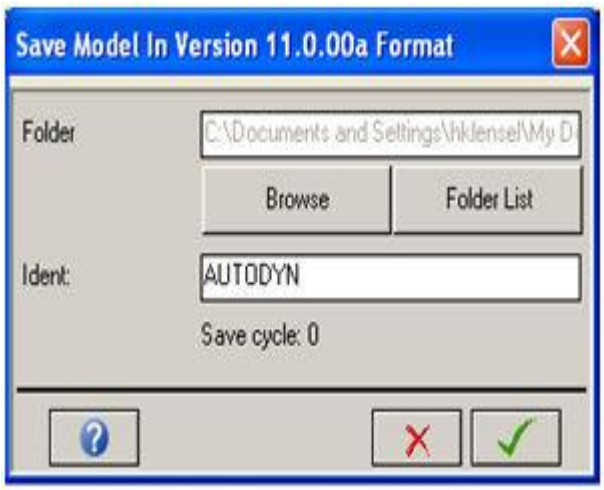
This option lets you to "Save" an AUTODYN save file in the format supported by previous versions. The formats available are listed and the chosen format should be selected from the list.

As features and options available in the current version, but not in the version selected, will be lost, we recommend that you always choose a new filename when using this option.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

PullDownMenus - File - Export To Version

This option lets you "Save" an AUTODYN save file in the format supported by previous versions. The formats available are listed and the chosen format should be selected from the list.

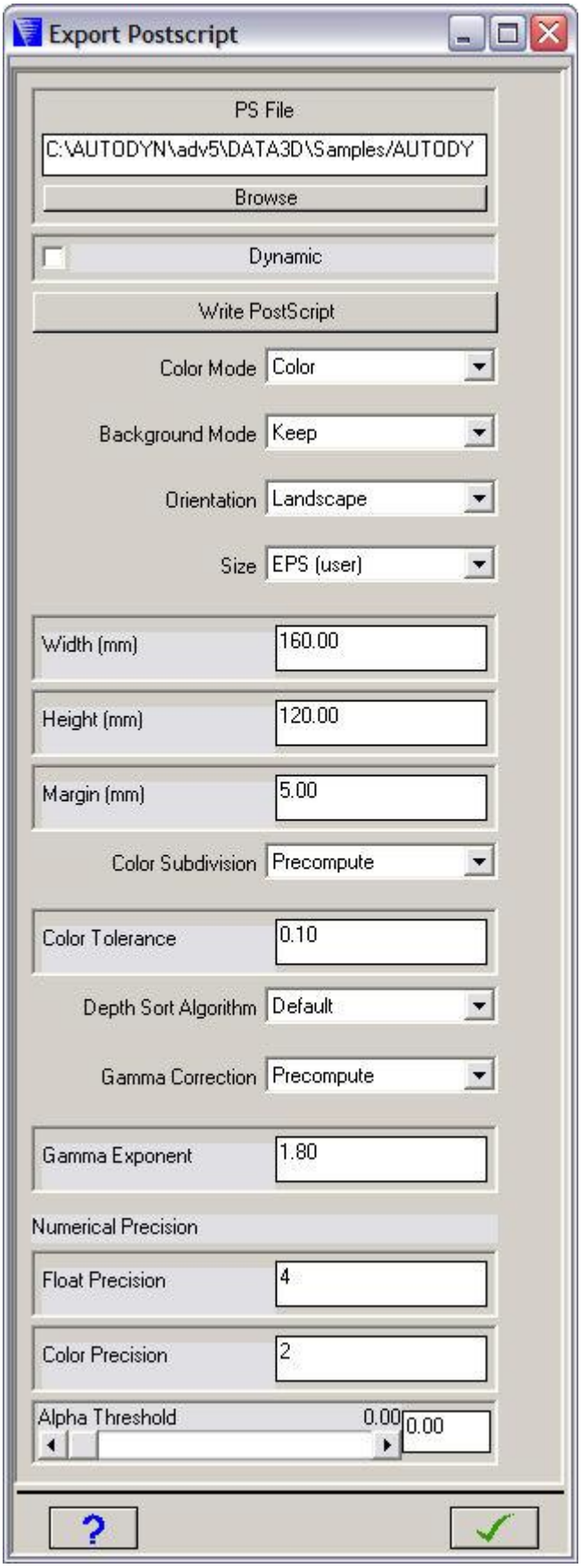


As features and options available in the current version, but not in the version selected, will be lost, we recommend that you always choose a new filename when using this option.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - File - Export PostScript

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



This window lets you output the current plot to a PostScript file.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

PS file

Enter the name of the PostScript file in this field (use the Browse button to navigate to a directory/file if you wish).

Dynamic

Check this box if you want a new PostScript file to be output each time the input view is rendered. Otherwise, you must explicitly request a new file by pressing the **Write PostScript** button.

Write PostScript

If **Dynamic** is not checked, press this button to generation of a PostScript output file.

Color Mode

This pull-down menu lets you select the color format used in the output file.

Color

Each color is specified as an RGB triple in the output file. The output file can be rendered in color on a color printer, or in greyscale on a monochrome printer.

Greyscale

Each color is converted to a single intensity value. The output file can only be rendered in greyscale, on color or monochrome printer. Greyscale files will be smaller than those written with full color information.

Background Mode

This pull-down menu lets you select how the background of the scene will be rendered.

White

Force the scene to be rendered on a white background. This typically means that the printer does not apply any color or shading to the background.

Black

Force the scene to be rendered on a black background.

Keep

Use the view background color specified in the AVS/Express viewer, as the PostScript background color.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Orientation

This pull-down menu lets you select the page layout.

Landscape

The x-axis of the scene device coordinates is oriented along the longer side of the paper.

Portrait

he x-axis of the scene device coordinates is oriented along the shorter side of the paper.

Size

This pull-down menu lets you select the paper size for the output.

EPS (user)

Encapsulated PostScript format, where you can specify the dimensions of the output, and the resulting file can be embedded in other documents. You must set the width, height and margin in the fields provided if you select this option.

A (letter)

Standard US letter paper size (280mm x 216mm)

B

Ledger paper size (432mm x 280mm).

A4

Standard European A4 paper size (297mm x 210mm).

A3

A3 paper size (420mm x 297mm).

A0

A0 paper size (1189mm x 841mm).

Width, Height, Margin

Use these fields to specify the width and height and margin of EPS size output if this option is selected.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Color Subdivision

This pull-down menu lets you select how the color is interpolated for polylines and surfaces, in 2D and 3D, with per-vertex color or lighting of 3D surfaces. When subdivision is enabled (**Precompute** or **Postcompute**), the subdivision proceeds until a color tolerance is reached. This adaptive method gives the best quality results without subdividing primitives with small color gradients.

None (flat)

No color interpolation is performed. The output polyline and triangle primitives are drawn with constant color per-segment, or per-facet. Incoming vertex colors are averaged to produce the segment or facet color, where necessary.

Precompute

The OutputVPS module subdivides line segments and triangles that have per-vertex color from data mapping, or as the result of lighting 3D surfaces. The subdivision generates many smaller primitives with linearly interpolated color. Precomputing the subdivision increases the output file size, and the subdivision cannot be changed during printing.

Postcompute

Special interpolation macros are written to the output file, together with the color tolerance parameter. Primitives are written with color per-vertex information in the PostScript output. The subdivision is made at print time, in the PostScript processor. This gives high quality output, with small file size, at the expense of slower printing. The color tolerance parameter can be edited by hand to change the color resolution after the file has been generated.

Color tolerance

Use this field to specify the absolute difference in color space, for the color subdivision algorithms: **Precompute** and **Postcompute**. In Postcompute mode, the color tolerance is written to the output file. In precompute mode it is used by the OutputVPS module to subdivide the line segments and triangle facets which have per-vertex color. This real value must be in the range 0.0 to 1.0. Smaller values signify a finer subdivision. Values close to zero will force a very large number of primitives to be generated, either precomputed in the output file (large file, slow printing), or postcomputed internally within the printer (slow printing). A value of 1.0 will not cause any subdivision to occur, even when it is enabled by the subdivision flag. The color tolerance user interface is not displayed for color subdivision None.

Depth Sort Algorithm

Select the depth sort algorithm you want to use from the pull-down menu.

Default

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

A simple sort by the minimum z coordinate of each primitive object. While this is well suited for many applications, it can result in visual artifacts in complex scenes.

NNS

The Newell and Sancha hidden surface removal algorithm. When a possible visual artifact is detected, the algorithm first tries to interchange the conflicting primitives. If there is still a conflict (consider the case of intersecting triangles), then one of the primitives is split along the
line(s) of intersection.

Gamma Correction

This pull-down menu lets you select how final colors are modified to take account of non-linear effects in the printing device, or perhaps to match the non-linear effects of the same scene displayed on a raster monitor.

None

No gamma correction is performed. Color values are assumed to behave linearly.

Precompute

The OutputVPS module modifies all color values (gray, red, green, blue), before writing the values to the file. Thus precomputed gamma correction is applied after precomputed color subdivision, but before postcomputed color subdivision. This means that postcomputed linear interpolation may not give the expected appearance. Precomputed gamma marginally slows file generation, but does not affect file size. The precomputed gamma factor cannot be changed after the file has been generated.

Postcompute (default)

The gamma exponent is written to the output file, and an exponential function is set for color transfer in the printer. Each rendered color is then gamma corrected at print time. Postcomputing the gamma correction only increases the file size by a few bytes, but it will slow down printing. The gamma factor can be edited by hand after the file has been generated.

Gamma exponent

Use this field to specify the gamma factor to be applied to color components, either precomputed in the output file, or postcomputed by the printer during printing. The gamma factor is an exponential factor applied to each color component independently (gray, red, green, blue). Its value must be greater than 0.0. A value of 1.0 leaves color unchanged, values between 0.0 and 1.0 darken the colors, and values greater than 1.0 lighten (desaturate) the colors. The default value is 1.8

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Numerical Precision

Float Precision

Use this field to specify the floating point precision used to write real values to the output file. The values include spatial coordinates and transformations, but not colors. The value must be in the range 1 to 8. The default value is 4

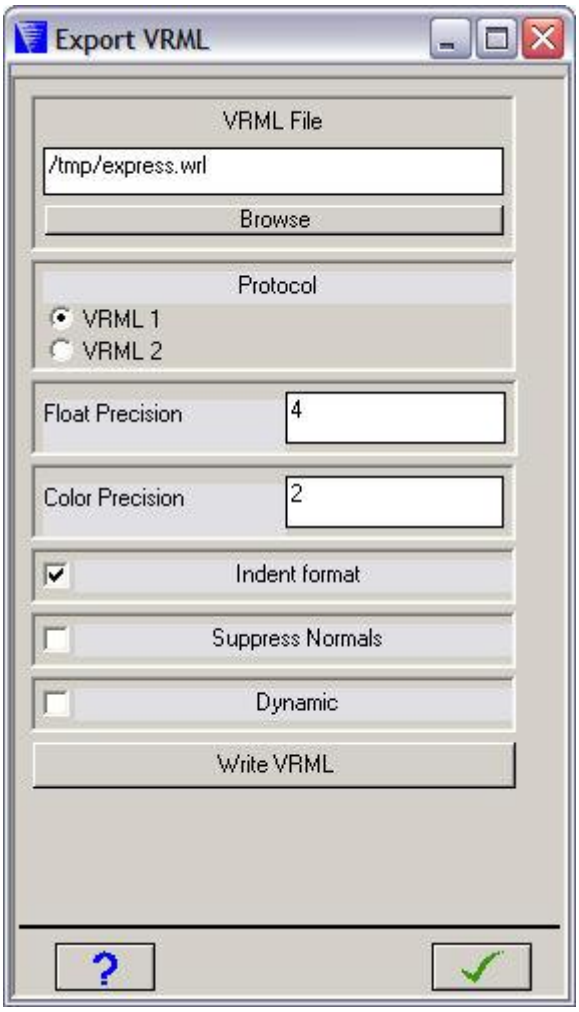
Color Precision

Use this field to specify the floating point precision used to write color values to the output file. The color values can be gray, red, green or blue. The value must be in the range 1 to 8.The default value is 2

Alpha Threshold

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - File - Export VRML



This window lets you output the current plot to a VRML file.

VRML file

Enter the name of the VRML file in this field (use the Browse button to navigate to a directory/file if you wish).

Protocol

Select the VRML protocol you want to use to write the output file.

VRML1

A clear text ASCII encoding of the VRML 1.0c standard (29 January 1996).

VRML2

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

A clear text UTF8 encoding of the VRML 2.0 standard, ISO/IEC CD 14772 (4 August 1996).

Float Precision

Use this field to specify the floating point precision when real values are written to the output file. The values include spatial coordinates, normal vectors, texture coordinates, material coefficients, and transformations, but not colors. The value must be in the range 1 to 8. The default value is 4.

Color Precision

Use this field to specify the floating point precision when real values are written to the output file. The values include diffuse material colors, emissive material colors, and light colors. The value must be in the range 1 to 8. The default value is 2.

Indent format

Check this box if you want the file to be written with indented nesting of VRML nodes. Indentation improves legibility, but increases the size of the output file.

Suppress Normals

Check this box if you do not want normals to be written to the VRML.

Dynamic

Check this box if you want a new VRML file to be output each time the input view is rendered. Otherwise, you must explicitly request a new file by pressing the **Write VRML** button.

Write VRML

When the dynamic switch is FALSE, this one-shot parameter triggers the generation of a VRML output file. When the dynamic switch is TRUE, the user interface for this parameter is removed.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Import

Pull-down Menu - Import

This pull-down menu lets you import grid data from other programs.

from TrueGrid (.zon)

Lets you import block zoning data from TrueGrid.

from ICEM (.geo)

Lets you import block zoning data from ICEM-CFD.

from LS-DYNA (.k)

Lets you import .k files from LS-DYNA.

from MSC.NASTRAN bdf (.dat)

Lets you import .bdf files from Nastran.

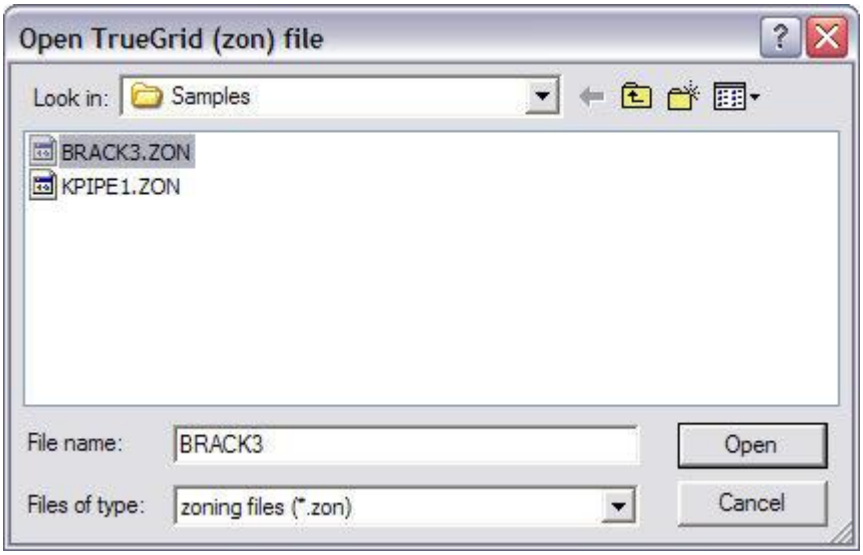
Convert IJK to Unstructured

Lets you convert existing IJK structured Parts to Unstructured Parts.

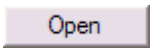
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

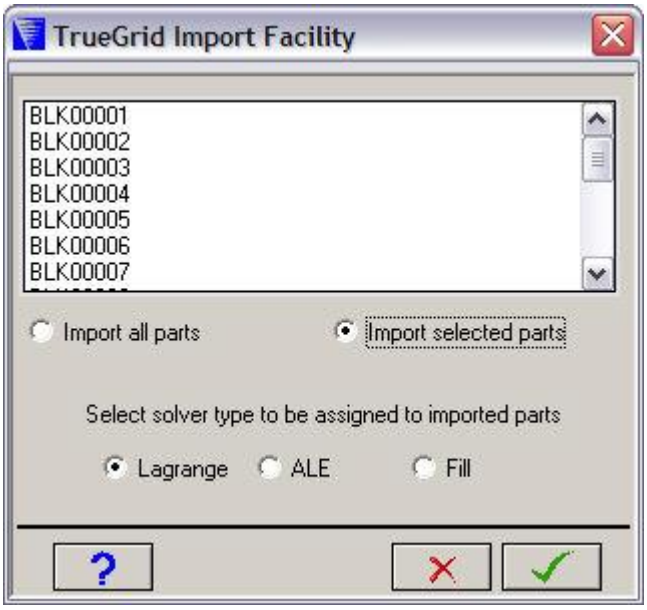
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Import - TrueGrid



This window lets you import grid data from TrueGrid.

Navigate to the TrueGrid zoning file you want to import (**.zon**) and press  to open the following window.



Blocks

The TrueGrid zoning file contains one or more blocks of nodes (imported as Parts). These are displayed in the scrolling box at the top of the window.

If Import selected parts is checked, you can select one or more blocks.

Import all Parts

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this option if you want to import all the blocks as Parts.

Import selected Parts

Check this option if you want to import selected blocks as Parts. Select the blocks you want to import in the top box.

Select solver type

Select the solver type you want to assign to the imported Parts.

Press to import the Parts.

Parts are imported with their block name. You can change these block names later in the Parts panel.

True*Grid* zoning file format

There are two formats of True*Grid* zoning files supported by AUTODYN - implicit and explicit. The two formats are illustrated below by an example which generates a single cell cubic part (IMAX=2, JMAX=2, KMAX=2) called SUB1 with corners (0.0,0.0,0.0) and (1.0,1.0,1,0) and a four cell cuboid part (IMAX=2, JMAX=2, KMAX=3) called SUB2 with corners (2.0,2.0,2.0) and (3.0,3.0,4.0).

The implicit form of this data would be as follows:

IMPLICIT	Indicates implicit input is to follow
SUB1	Name of the part for which data follows
1 2 1 2 1 2	The (I,J,K) range for the data I=1, 2, J=1,2, K=1,2
0.0 0.0 0.0	(x,y,z) coordinates for node with index (1,1,1)
0.0 0.0 1.0	(x,y,z) coordinates for node with index (1,1,2)
0.0 1.0 0.0	(x,y,z) coordinates for node with index (1,2,1)
0.0 1.0 1.0	(x,y,z) coordinates for node with index (1,2,2)
1.0 0.0 0.0	(x,y,z) coordinates for node with index (2,1,1)
1.0 0.0 1.0	(x,y,z) coordinates for node with index (2,1,2)
1.0 1.0 0.0	(x,y,z) coordinates for node with index (2,2,1)
1.0 1.0 1.0	(x,y,z) coordinates for node with index (2,2,2)
SUB2	Name of the part for which data follows
1 2 1 2 1 3	The (I, J, K) range for the data I=J, 2, J=1, 2, K=1, 3
2.0 2.0 2.0	(x,y,z) coordinates for node with index (1,1,1)
2.0 2.0 3.0	(x,y,z) coordinates for node with index (1,1,2)
2.0 2.0 4.0	(x,y,z) coordinates for node with index (1,1,3)
2.0 3.0 3.0	(x,y,z) coordinates for node with index (1,2,1)
2.0 3.0 4.0	(x,y,z) coordinates for node with index (1.2,2)
3.0 2.0 2.0	(x,y,z) coordinates for node with index (2,1,1)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

3.0 2.0 3.0	(x,y,z) coordinates for node with index (2,1,2)
3.0 2.0 4.0	(x,y,z) coordinates for node with index (2,1,3)
3.0 3.0 2.0	(x,y,z) coordinates for node with index (2,2,1)
3.0 3.0 3.0	(x,y,z) coordinates for node with index (2.2,2)
3.0 3.0 4.0	(x,y,z) coordinates for node with index (2,2,3)
END	

The explicit form of this data would be as follows:

EXPLICIT	Indicates explicit input is to follow
SUB1	Name of the part for which data follows
1 1 1 0.0 0.0 0.0	(x,y,z) coordinates for (1,1,1)
1 1 2 0.0 0.0 1.0	(x,y,z) coordinates for (1,1,2)
1 2 1 0.0 1.0 0.0	(x,y,z) coordinates for (1,2,1)
1 2 2 0.0 1.0 1.0	(x,y,z) coordinates for (1,2,2)
2 1 1 1.0 0.0 0.0	(x,y,z) coordinates for (2,1,1)
2 1 2 1.0 0.0 1.0	(x,y,z) coordinates for (2,1,2)
2 2 1 1.0 1.0 0.0	(x,y,z) coordinates for (2,2,1)
2 2 2 1.0 1.0 1.0	(x,y,z) coordinates for (2,2,2)
SUB2	Name of the part for which data follows
1 1 1 2.0 2.0 2.0	(x,y,z) coordinates for (1,1,1)
1 1 2 2.0 2.0 3.0	(x,y,z) coordinates for (1,1,2)
1 1 3 2.0 2.0 4.0	(x,y,z) coordinates for (1,1,3)
1 2 1 2.0 3.0 2.0	(x,y,z) coordinates for (1,2,1)
1 2 2 2.0 3.0 3.0	(x,y,z) coordinates for (1.2,2)
1 2 3 2.0 3.0 4.0	(x,y,z) coordinates for (1,2,3)
2 1 1 3.0 2.0 2.0	(x,y,z) coordinates for (2.1,1)
2 1 2 3.0 2.0 3.0	(x,y,z) coordinates for (2,1,2)
2 1 3 3.0 2.0 4.0	(x,y,z) coordinates for (2,1,3)
2 2 1 3.0 3.0 2.0	(x,y,z) coordinates for (2,2,1)
2 2 2 3.0 3.0 3.0	(x,y,z) coordinates for (2.2,2)
2 2 3 3.0 3.0 4.0	(x,y,z) coordinates for (2,2,3)
STOP	Indicates end of all data

With the explicit format, it is not necessary to define coordinates for a complete block of nodes in index space, and the order in which nodes are defined is unimportant.

When this option is used for a specific block of nodes in index space, the "indent.zon" file is scanned and only those nodes defined in this file and within the specified part and block to be initialized will be assigned coordinate values. Thus, standard zoning options could be used to define the bulk of the grid and the "User" option called to redefine the coordinates of a few isolated nodes.

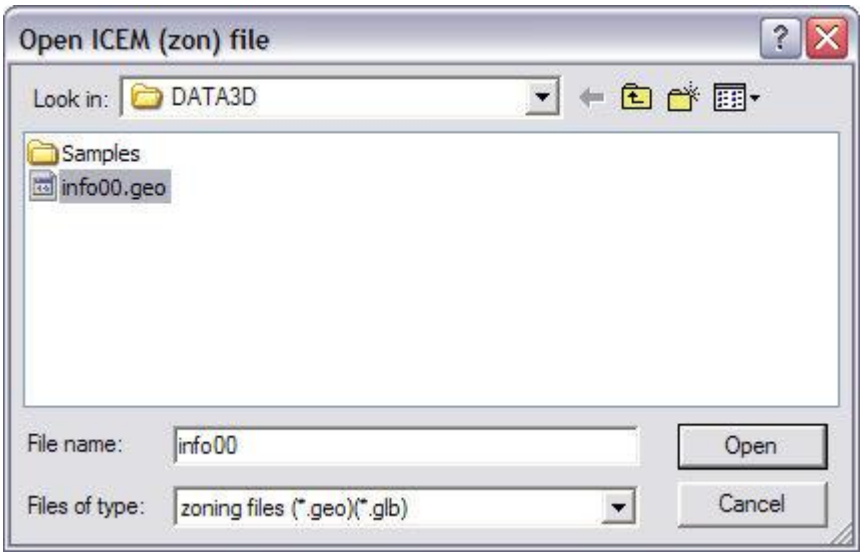
Before reading in an explicit zoning file, the parts must already be created. The file can then be read in using the Import button on the Parts Zoning panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The above example is for 3D. For 2D models the format is the same except of course there is no K index or Z coordinates.

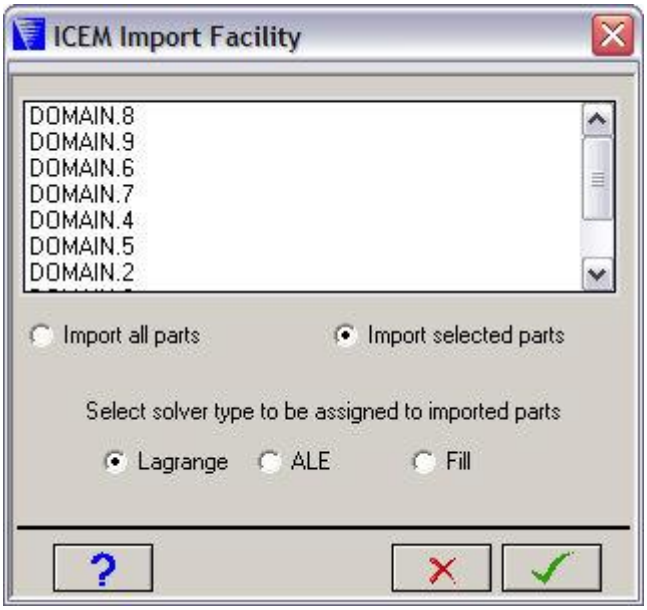
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Import - ICEM



This window lets you import grid data from ICEM-CFD.

Navigate to the ICEM-CFD zoning file you want to import (**.geo**) and press Open to open the following window.



Domains

The ICEM-CFD zoning file contains one or more domains of nodes (imported as Parts). These are displayed in the scrolling box at the top of the window.

If Import selected parts is checked, you can select one or more domains.

Import all Parts

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this option if you want to import all the domains as Parts.

Import selected Parts

Check this option if you want to import selected domains as Parts. Select the domains you want to import in the top box.

Select solver type

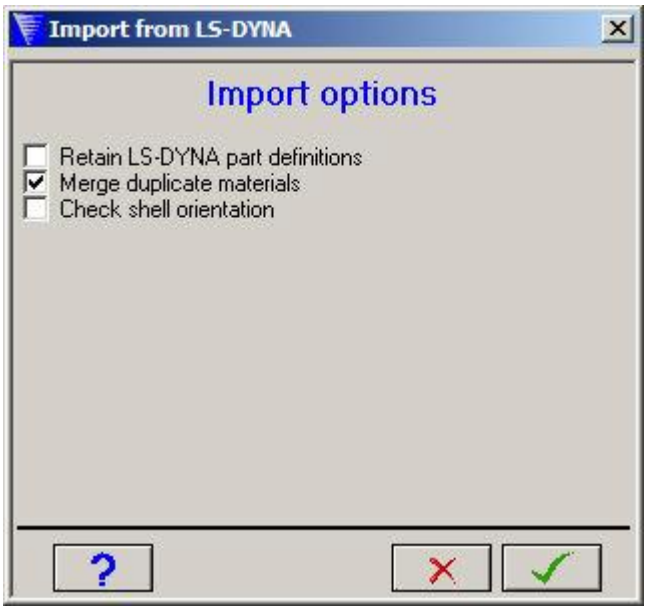
Select the solver type you want to assign to the imported Parts.

Press to import the Parts.

Parts are imported with their domain name. You can change these domain names later in the Parts panel..

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Import - LS-DYNA



Import Options

After browsing for an LS-DYNA keyword file, select import options in this dialog.

Retain LS-DYNA part definitions

Select this option to name the important parts from definitions within the LS-DYNA file. Otherwise, generic names are automatically generated for the parts.

Merge duplicate materials

Select this option to merge duplicate materials

Check Shell orientation

Select this option to check for inconsistent or non-manifold shell meshes. If the mesh is inconsistent and manifold, you should re-orient the mesh to have consistent normals.

LS-DYNA Keyword File Format

Both formatted and free formatted (comma delimited) LS-DYNA keyword files can be imported into AUTODYN. The standard used is based on version 970 of LS-DYNA.

The primary use of the LS-DYNA reader is to allow you to define a mesh to be used with the unstructured solvers. In addition, other information regarding material definitions, boundary conditions and initial conditions can be imported and stored in AUTODYN.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

An overview of the information, which can be imported into AUTODYN from an LS-DYNA keyword file is given below.

Mesh

The mesh, represented as nodal positions and element connectivity, is read into AUTODYN and sorted to produce Parts and Components. A Part is a collection of connected elements of the same classification e.g. volume elements, shell elements and beam elements. A Component is a collection of all connected elements regardless of their topology. By default, the defined Parts in AUTODYN will not necessarily be equivalent to those defined in the LS-DYNA keyword file.

An option to retain the Part definitions in the LS-DYNA file is also available.

Materials

Each element defined in the keyword file will have an associated material definition. Corresponding materials will be automatically generated during the import of the keyword file into AUTODYN. In general (except for the materials identified below), the actual material parameters will not be read into AUTODYN and you should ensure that all materials are properly defined after the import.

At present, a subset of the material definitions and parameters are automatically read into AUTODYN. These include materials of type rigid, elastic, piecewise linear plasticity and spring elastic material models.

Initial and Boundary Conditions

Fixed and prescribed boundary conditions are imported from the LS-DYNA keyword file into AUTODYN. These are converted into the equivalent AUTODYN boundary conditions including parameters, where possible.

Initial velocities defined on nodes and sets of nodes in the keyword file are read into AUTODYN and converted into Initial Conditions. The initial condition is also automatically applied to the identified nodes.

Shell and Beam Sections

Shell sections defined in the keyword file are automatically transformed into AUTODYN initial conditions. The local direction for layered shells is not currently translated.

Beam cross-sections and directions are read from the keyword file and these and the relevant data are converted to be compatible with beam types used in AUTODYN.

Keyword Compatibility

The current capabilities of the AUTODYN import capability for LS-DYNA (.k) keyword files is summarized below:

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Keyword	Compatibility
*BOUNDARY_SPC_NODE	Translational velocities only. DORFX, DORFY, DORFZ not recognized
*BOUNDARY_SPC_SET	Translational velocities only. DORFX, DORFY, DORFZ not recognized
*DEFINE_CURVE	Full
*ELEMENT_SOLID	Full
*ELEMENT_SHELL	Full
*ELEMENT_BEAM	Full
*INITIAL_VELOCITY	BOXID and IRIGID are not recognized
*INITIAL_VELOCITY_NODE	Translational velocities only. VXR, VYR and VZR are not recognized.
*INITIAL_VELOCITY _GENERATION	Translational velocities only. OMEGA, XC, YC, ZC, NX, NY, NZ and PHASE not recognized.
*MAT_PIECEWISE_LINEAR _PLASTICITY	Full
*MAT_ELASTIC	Full
*MAT_RIGID	Full
*MAT_SPRING_ELASTIC	Full
*NODE	Full
*PART	Full
*SECTION_BEAM	Beam types 1-5 supported. QR/IRID, SCOOR, NSM, NSLOC, NTLOC and IRR are not recognized.
*SECTION_SHELL	Shell thickness at first node is read in and set a shell thickness in initial condition.
*SET_NODE_LIST	Full
*SET_PART_LIST	Full
*SET_SHELL_LIST	Full
*SET_BEAM	Full
*SET_DESCRETE	Full
*SET_SOLID	Full
*SET_TSHELL	Full
*SET_NODE_LIST_GENERATE	Full
*SET_PART_LIST_GENERATE	Full
*SET_SHELL_LIST_GENERATE	Full
*SET_BEAM_GENERATE	Full
*SET_DISCRETE_GENERATE	Full
*SET_SOLID_GENERATE	Full

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

*SET_TSHELL_GENERATE	Full

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Import - Nastran

Import Options

After browsing a NASTRAN input file, select import options in this dialog.

Convert Properties to AUTODYN parts

Select this option to name the imported parts from definitions within the NASTRAN file. Otherwise, generic names are automatically generated for the parts.

Merge duplicate materials

Select this option to merge duplicate materials.

Check shell orientation is consistent

Select this option to check for inconsistent or non-manifold shell meshes. If the mesh is inconsistent and manifold, you should re-orient the mesh to have consistent normals.

Input File Format

AUTODYN can import MSC.NASTRAN™ input files in both free and fixed format. In free format, the entries can appear anywhere on the line and are separated by commas. With fixed format, a field must be located in a set part of the line. The fixed format field can be small (8 characters) or large (16 characters).

In addition to the MSC.NASTRAN™ bulk data input, AUTODYN can recognize the expanded input data set for explicit transient dynamics, as given by MSC.DYTRAN™ version 4.7.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The primary use of the MSC.NASTRAN™ reader is to allow you to define a mesh to be used with the unstructured solvers. In addition, other information regarding material definitions, boundary conditions and initial conditions can be imported and stored in AUTODYN.

An overview of the information, which can be imported into AUTODYN from an MSC.NASTRAN™ bulk data file is given below.

Mesh

The mesh, represented as nodal positions and element connectivity, is read into AUTODYN and sorted to produce Parts and Components. A part is a collection of connected elements of the same classification e.g. volume elements, shell elements and beam elements. A component is a collection of all connected elements regardless of their topology.

Materials

Each element, imported from the bulk data file will have the material name as defined on the property field for the element (PSOLID, PSHELL). Corresponding materials will be automatically generated during the import of the keyword file into AUTODYN. In general (except for the materials identified below), the actual material parameters will not be read into AUTODYN and you should ensure that all materials are properly defined after the import.

At present, a subset of the material definitions and parameters are automatically read into AUTODYN. These include materials of type rigid, elastic, piecewise linear plasticity and spring elastic material models.

Initial and Boundary Conditions

Every element property is translated into an Initial Condition in AUTODYN and allocated to the appropriate elements.

Fixed and prescribed boundary conditions are imported from the bulk data file into AUTODYN. These are converted into the equivalent AUTODYN boundary conditions including parameters, where possible.

Initial velocities defined on nodes and sets of nodes in the bulk data file are read into AUTODYN and converted into Initial Conditions. The initial condition is also automatically applied to the identified nodes.

Bulk Data Compatibility

The current capabilities of the AUTODYN import capability for MSC.NASTRAN™ bulk data files is summarized below:

Keyword	Compatibility
BEGIN BULKDATA	Full
CBAR	Only grid numbers to define local x-

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

	y plane
CBEAM	Only grid numbers to define local x-y plane
CHEXA	Full
CPENTA	Full
CQUAD4	No material direction
CTETRA	Full
CTRIA3	No material direction
JOIN	Full
FORCE	Full
GRID	No coordinate system; no SPC
PBAR	Full
PBEAM	Only constant cross-section
PBEAM1	Only as PBEAM
PCOMP	Full
PLOAD	Full
PSHELL	No thickness
PSHELL1	Up to field 7
PSOLID	Full
SET1	Full 1
SPC	Full
SPC1	No ranges
TABLED1	Full
TLOAD1	Only FORCE

Any keyword not mentioned in the table above will be ignored.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull Down Menus - Import - Convert IJK to Unstructured

Existing structured (IJK) Parts using the Lagrange, Shell and Beam solvers can be converted into Unstructured Parts via the Import, Convert IJK Part to Unstructured option. The mesh, material, boundary conditions and initial conditions will be transferred to the new Part.

This feature lets you convert existing models to the new solvers to take advantage of the reduced memory requirements and increased speed of solution.

During the conversion, you will be presented with the options

Conversion options
Delete converted IJK regions
Merge connected IJK Parts into single Unstructured Part
Merge joined IJK nodes into single Unstructured node

Delete converted IJK regions
This option deletes the structured IJK Parts that have been converted to unstructured solvers after the conversion process.

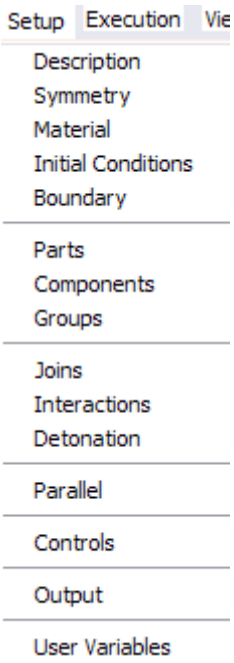
Merge connected IJK Parts into single Unstructured Part
Complex geometries defined using the structured (IJK) solvers tend to be made up of a large number of relatively simple Parts that are joined together (for example a mesh imported from ICEM-CFD .geo file). By default, the conversion process will merge connected IJK Parts into a single Part. This simplifies the model and can also improve performance because there will be fewer Joins in the model. For some applications, you may not wish to merge the IJK Parts in this way. De-selecting this option will result in no change in the Parts defined in the model.

Merge joined IJK nodes into single unstructured node
By default, all joined nodes are merged into a single unstructured node during the conversion process. In some cases, for example if you have pinned joins, you may not wish to do this.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Setup

Pull-down Menu - Setup



This pull-down menu lets you set up models. Most of these options are accessible through the Navigation Bar.

Description

Lets you modify the description of your model.

Symmetry

Lets you change the symmetry settings for your problem.

Material

Lets you define and modify materials.

Initial Conditions

Lets you define and modify Initial Condition Sets
.

Boundary

Lets you define and modify Boundary Conditions.

Parts

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Lets you define and modify Parts

Components

Lets you define and modify Components

Groups

Lets you define and modify Groups

Joins

Lets you define joins.

Interactions

Lets you define interactions

Detonation

Lets you define detonation points.

Parallel

Lets you set up your parallel processing environment and prepare your model for parallel processing.

Controls

Lets you set various controls for your model such as run limits, timestep controls, damping constants, cutoffs and gravitational forces.

Output

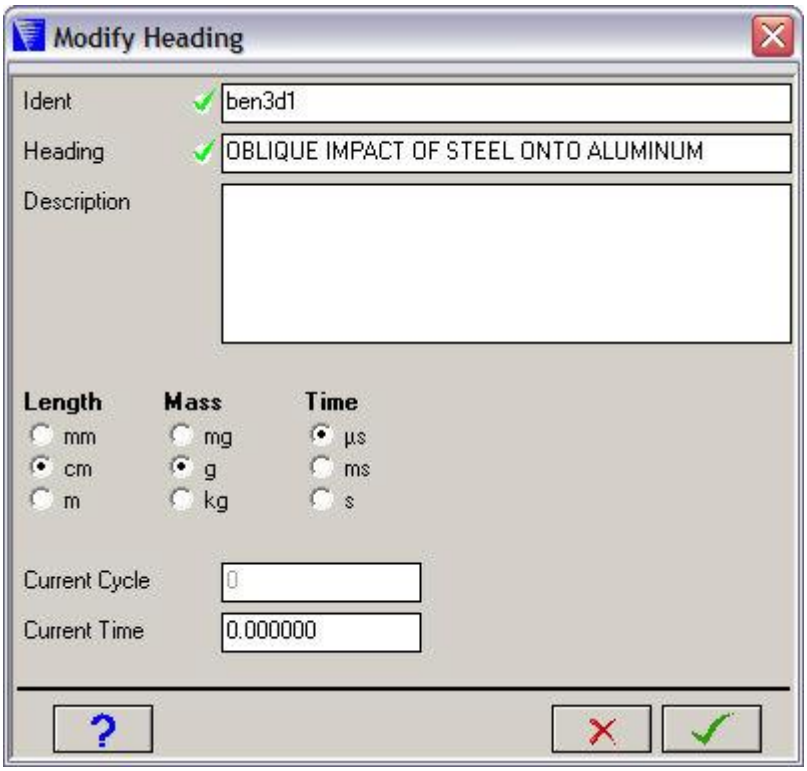
Lets you select the output you want from your model.

User Variables

Lets you initialize any grid variables you want to use in your user subroutines.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Setup - Description



This window lets you modify the description of your model.

Ident

This field lets you change the ident for your model.

The ident you choose is used to identify all the files created for your model.

e.g. If your Ident is **MyImpactProblem**, then the cycles zero Save file for the problem will be named **MyImpactProblem_0.ad.**

Heading

This field lets you change the heading for your model.

Headings appear in plots and other output for your model.

Description

This field lets you enter or change a detailed description of your model.

Units

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This field lets you change the units of length, mass and time you want to use for your model.

The default units (mm,mg,ms) will give good results, with small rounding errors, for most models.

Changing the units here **WILL NOT** result in any scaling of data previously entered. It only effects the units displayed in output and data conversions performed when reading / writing data to / from material libraries.

Current Cycle

This field displays the current cycle number.

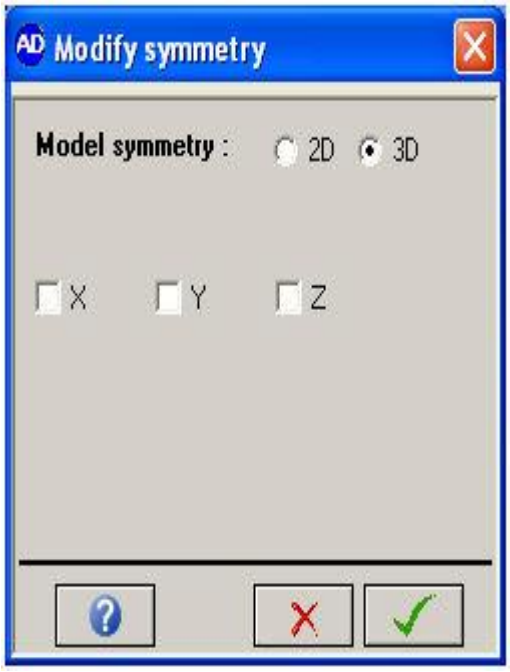
Current Time

This field lets you change the current time.

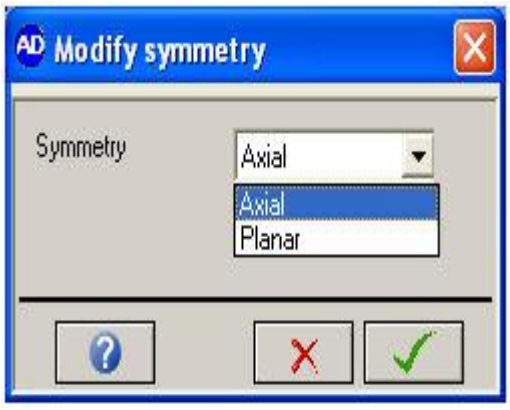
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Setup - Symmetry

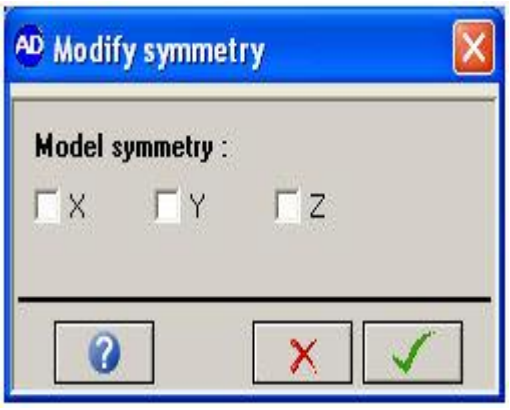
AUTODYN started up in Workbench



AUTODYN started up outside from ANSYS Workbench



2D Window



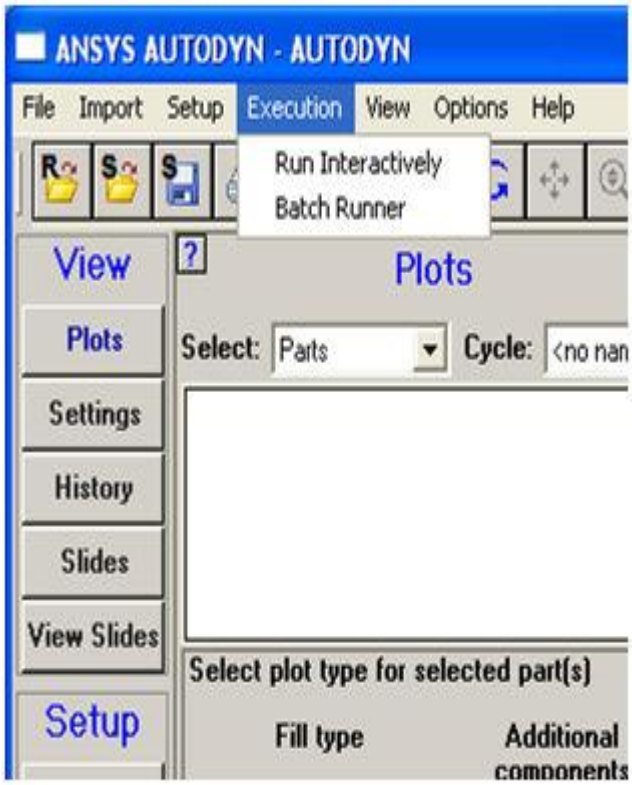
3D Window

This window lets you change the symmetry used in you model.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Execution

Pull-down Menu - Execution



This pull-down menu lets you execute your calculation.

Run Interactively

Lets you execute you problem interactively. This is the same as pressing

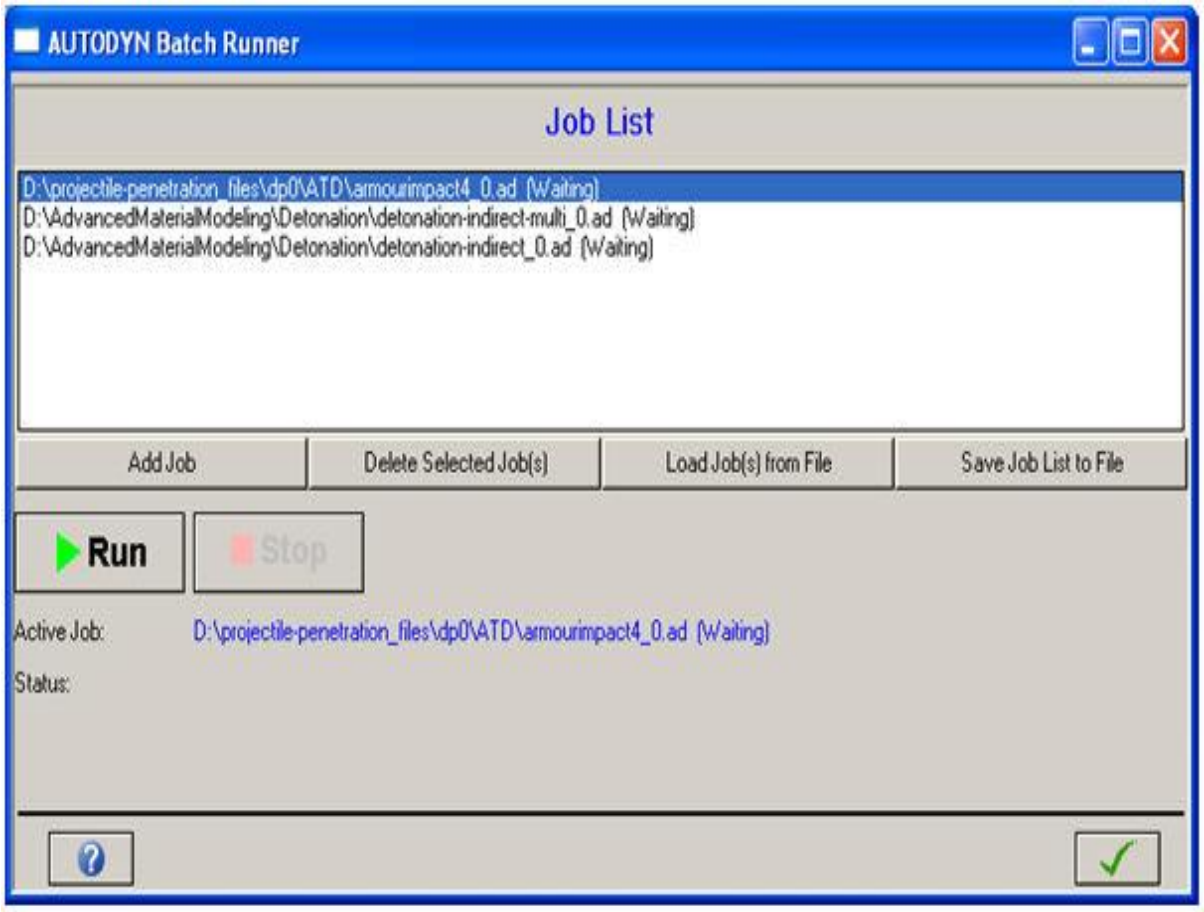
on the Navigation bar.

Batch Runner

Lets you run one or more of your models in batch mode (i.e. without graphics).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Execute - Batch Runner



This Window lets you set up and run single or multiple Jobs (calculations) in batch mode (without graphics).

Job List

This box lists all current jobs (Save files to be loaded and executed) and shows their status.

Add Job

Press this button to add a job to the list. A window opens to let you browse to the Save file you want to load and execute.

Delete Selected Job(s)

Select one or more Jobs in the Job List and press this button to delete them from the Job List.

Load Job(s) from File

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

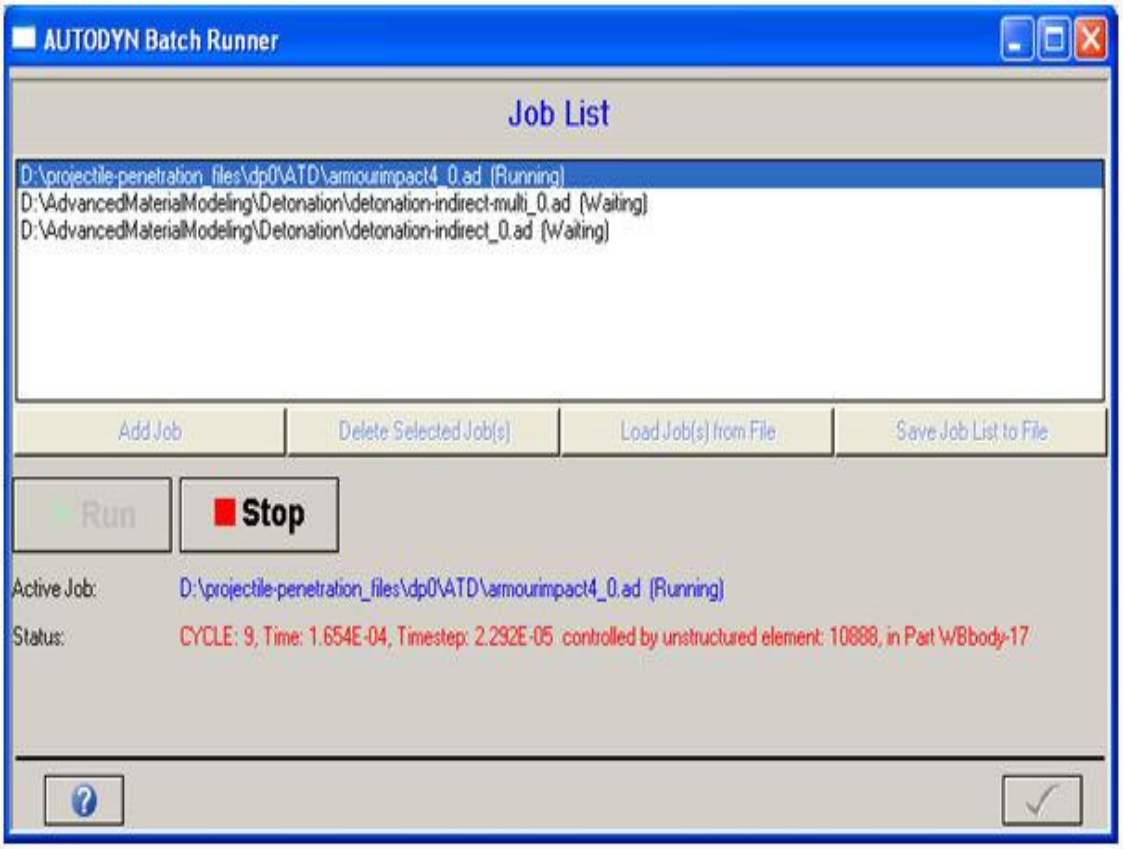
Press this button to load jobs saved in a file (**.bat**) into the Job List.

Save Job List to File

Press this button to save all the jobs in the current Job List to a file (**.bat**).

Run

Press this button to run the jobs in your Job List.



Jobs run sequentially in the Job List.

While running, the window displays information on which jobs have completed, which job is currently executing and which are waiting in line to execute. The status of the executing job is also displayed at the foot of the window.

Stop

Press this button to stop running jobs in the Job List.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

View

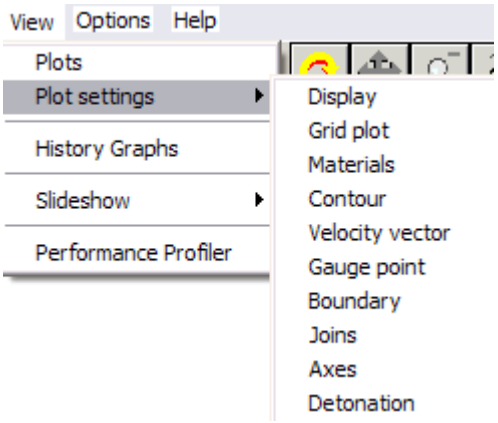
Pull-down Menu - View

This pull-down menu lets you customize your view (what you see in the View panel). Most of these options are accessible through the Navigation Bar.

Plots

Lets you set up what you want to see in the View panel.

Plot Settings



Lets you change settings for the various plot types that control how you view your model.

You can change settings for the following plot types:

Display
Grid
Materials
Contour

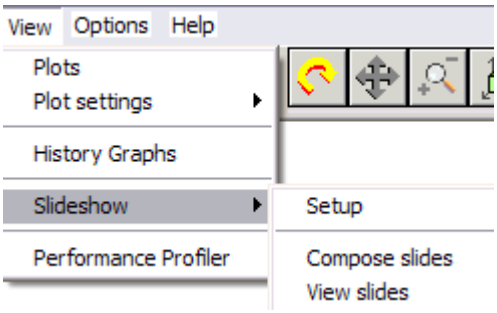
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Velocity Vector
Gauge Point
Boundary
Joins
Axes
Detonation

History Graphs

Lets you view histories of Gauge points and Material, Part and Energy summaries.

Slideshow



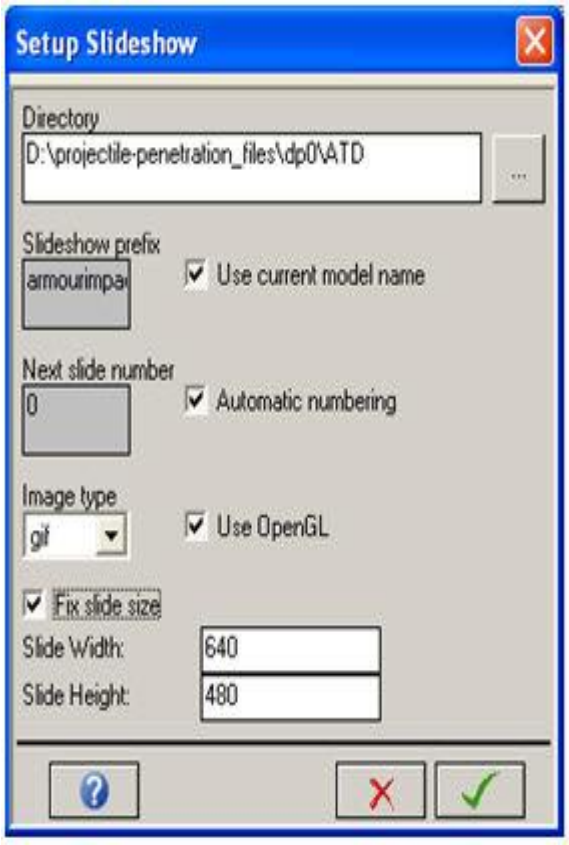
Lets you create and organize slideshows (animations)

Performance Profiler

Brings up a performance profiler that lets you monitor your calculation.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - View - Slideshow - Setup



This window lets you set parameters for creating a slideshow.

Directory

Press next to the Directory box to browse to the directory in which you want to save your slideshow.

Slideshow prefix

Enter a slideshow prefix in this field or check **Use current model name** to use the current Ident as the slideshow prefix.

This prefix is used for all image files created for the slideshow.

Next slide number

Enter a number for the next slide to be created or check **Automatic numbering** if you want AUTODYN to automatically assign numbers to slides.

Image type

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Use this pull-down menu to select the image type you want to use to create your slides (gif, jpeg, tiff or gfa).

Use OpenGL

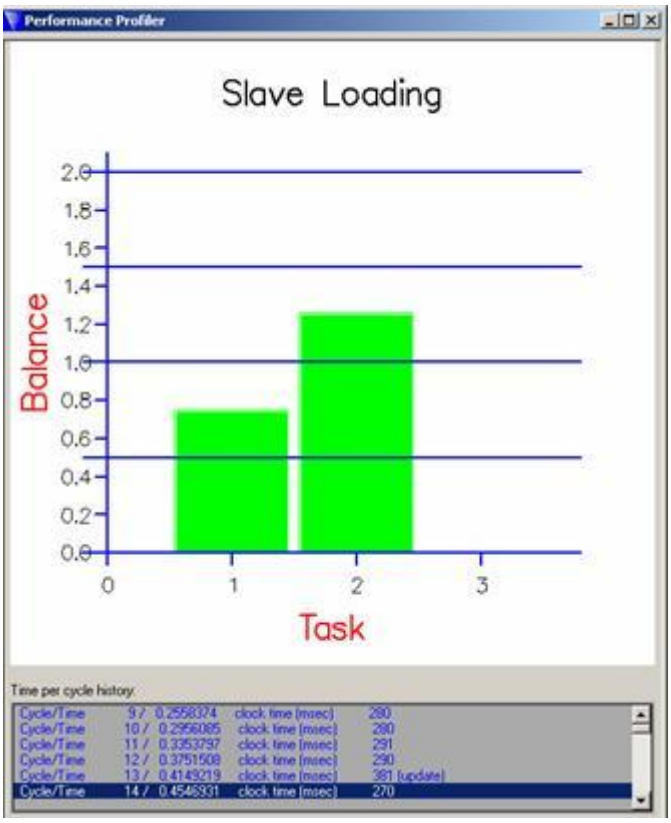
Check this box if you want to use OpenGL to create your slides. This can significantly improve the plotting of SPH objects.

Fix Slide size

Select this toggle to fix the size of slides created. Specify the width and height of the image in pixels. If the toggle is not selected, the slides created are to be the size of the graphics window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - View - Performance Profiler



This window provides a Performance Profiler that lets you visually monitor the load balancing performance of parallel simulations.

To take full advantage of this option, you must create an empty file **profile.cfg** in the directory where the AUTODYN model resides.

If you do this, the following type of output will be displayed in the scrolling box at the foot of the window:

```
Cycle/Time      1  /  1.637809225157053E-4 clock time (msec) 32856  (update)
Cycle/Time      2  /  3.439399411878227E-4 clock time (msec) 17030
Cycle/Time      3  /  5.421148660224775E-4 clock time (msec) 17056
Cycle/Time      4  /  7.601072880654561E-4 clock time (msec) 17029
Cycle/Time      5  /  9.998989575100770E-4 clock time (msec) 17054
Cycle/Time      6  /  1.263669799616239E-3 clock time (msec) 17016
Cycle/Time      7  /  1.553817732221804E-3 clock time (msec) 17081
Cycle/Time      8  /  1.872980465005591E-3 clock time (msec) 16994
Cycle/Time      9  /  2.224059478677190E-3 clock time (msec) 17116
Cycle/Time     10  /  2.610246402086324E-3 clock time (msec) 17024
Cycle/Time     11  /  3.035052027043786E-3 clock time (msec) 17075
Cycle/Time     12  /  3.502338224625149E-3 clock time (msec) 17070
Cycle/Time     13  /  4.016353053105620E-3 clock time (msec) 17053
Cycle/Time     14  /  4.581769376689206E-3 clock time (msec) 17019
Cycle/Time     15  /  5.203727346111727E-3 clock time (msec) 17070
```


Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

```
Cycle/Time    16  /  5.887881127305133E-3 clock time (msec) 17011
Cycle/Time    17  /  6.640450302929377E-3 clock time (msec) 17085
Cycle/Time    18  /  7.468276414058693E-3 clock time (msec) 34853  (update)
Cycle/Time    19  /  8.378885156037852E-3 clock time (msec) 17017
Cycle/Time    20  /  9.380554793925532E-3 clock time (msec) 17098
```

Each line gives the cycle number, the cycle time, and the wall clock time that the calculation used to compute the cycle (in milliseconds). If the line ends with "**(update)**", this indicates that work unit information for the interaction logic was updated on that cycle. Such cycles usually take longer to compute.

The profiler also shows you visually how well balanced your calculation is. In the example shown above, **Balance** is a measure of the relative CPU time spent on each task for each cycle in the simulation. So a balance of 1.0 for each task is the ideal / most efficient goal. A balance of less than 1.0 means that a task is under utilized (does not have enough work to do). A balance of greater than 1.0 means that a task is over utilized (has too much work to do).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Options

Pull-down Menu - Options

This pull-down menu lets you set various GUI options.

Interactivity

Lets you set how the mouse and keyboard is used for the Tool Bar operations that manipulate images in the view panel.

Clipping Planes

Lets you set clipping planes for your plot.

Default Settings

Lets you change the default settings used to start up AUTODYN.

Pause License

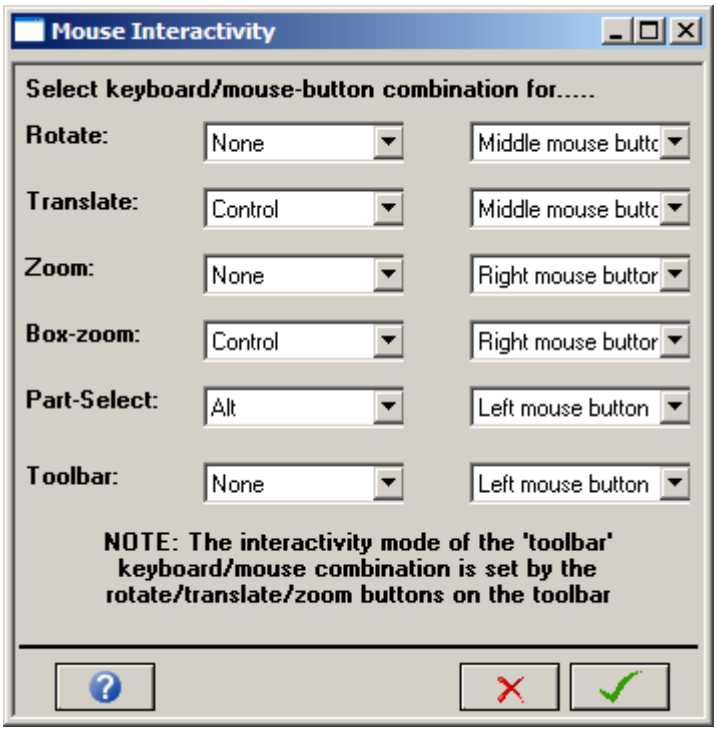
Lets you temporarily pause the license and free it for another AUTODYN instance.

Create User-Subroutine Project

Lets you install the AUTODYN project to create your own version of AUTODYN by using user-subroutines.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

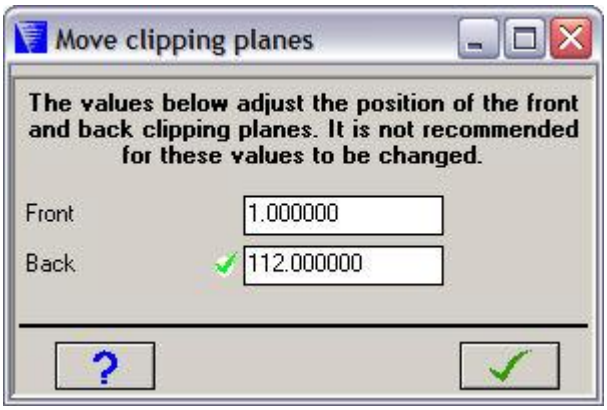
Pull-down Menu - Options - Interactivity



This window lets you change how the mouse and keyboard is used for the Tool Bar operations that manipulate images in the view panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Options - Clipping Planes


Move clipping planes
The values below adjust the position of the front and back clipping planes. It is not recommended for these values to be changed.
Front
1.000000
Back
112.000000
?

This window lets you set clipping planes for your plot.

If you zoom in close to, or into a grid, the image will not look good if you include everything in your view. The default values for the clipping planes usually improve such plots considerably, but there may be times when you want to use other values.

Front

Use this field to set the distance of the front clipping plane from the viewpoint. Any part of the grid nearer to the viewpoint than this plane will not be plotted.

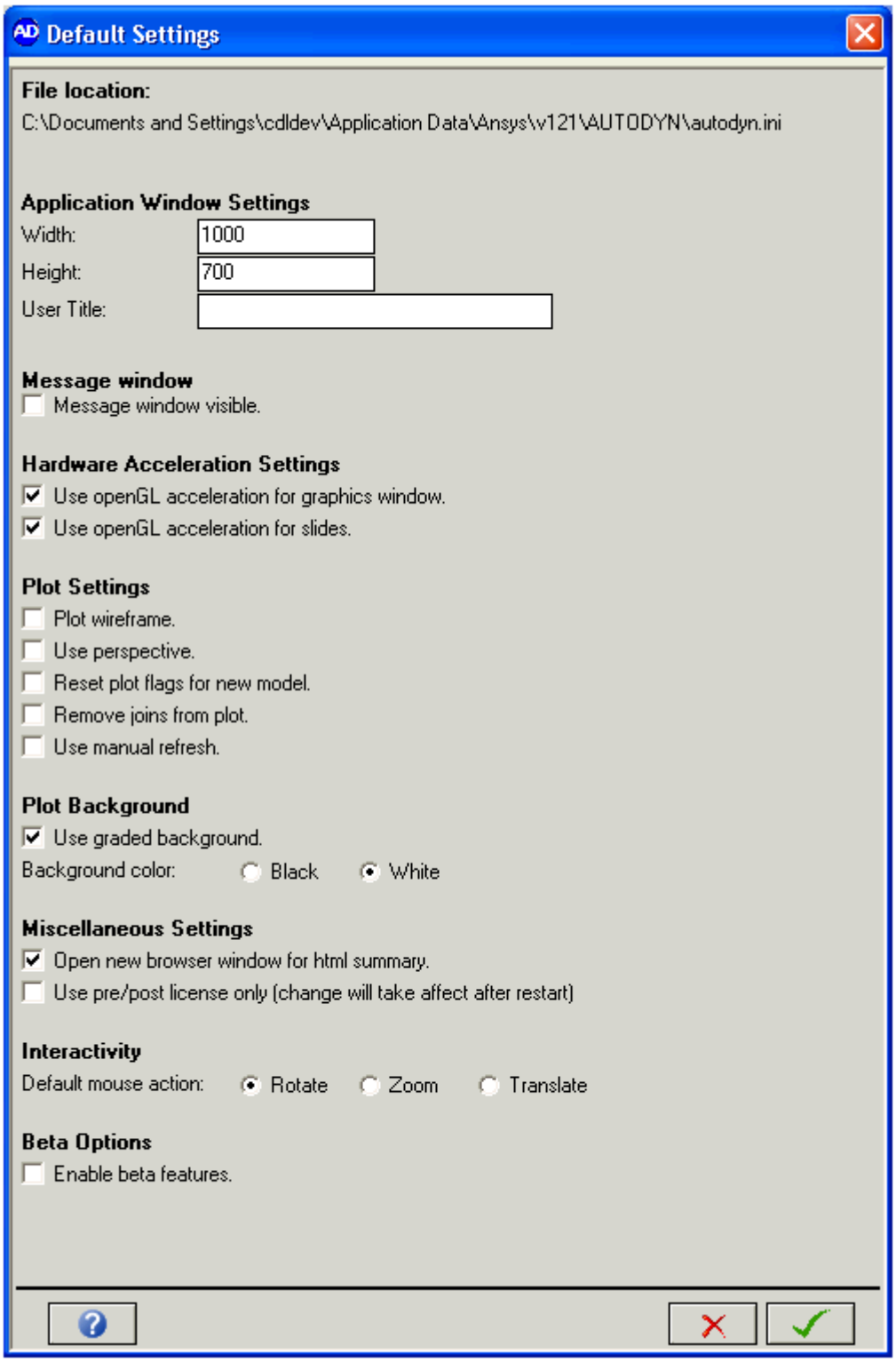
Back

Use this field to set the distance of the back clipping plane from the viewpoint. Any part of the grid further away from the viewpoint than this plane will not be plotted.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Options - Default Settings

This dialog lets you to define the default settings for the AUTODYN GUI. These settings are stored in an ASCII file "autodyn.ini" and are used to initialize various aspects of the GUI and rendering options, each time you start AUTODYN.

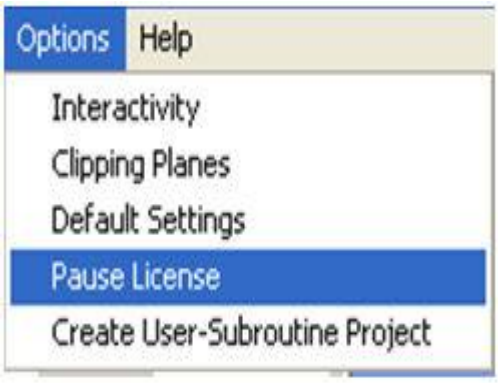


Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Options - Pause License

If you are working with a project in Workbench which contains an AUTODYN system linked to an ANSYS Explicit Dynamics system you will not be able to open and work in both applications simultaneously in shared licensing mode or when the license server only has one solver license available. For example, if you want to generate a new mesh in the meshing application then you will need to close the AUTODYN application to release the license for the meshing application to use.

As an alternative the user can leave the AUTODYN application open, but release the license for another application to use. While the license is released, the AUTODYN user interface will be locked.



After selection of the Pause option the license is released, and the AUTODYN user interface will be locked. To continue the AUTODYN application the user can select Retry to try to obtain the license again. The Close App option will exit the AUTODYN application without saving any data.

Warning!
This editor is PAUSED because the license is currently in use by another editor in this Workbench session.
All commands to this editor will be ignored until the license can be obtained.
Retry
Close App

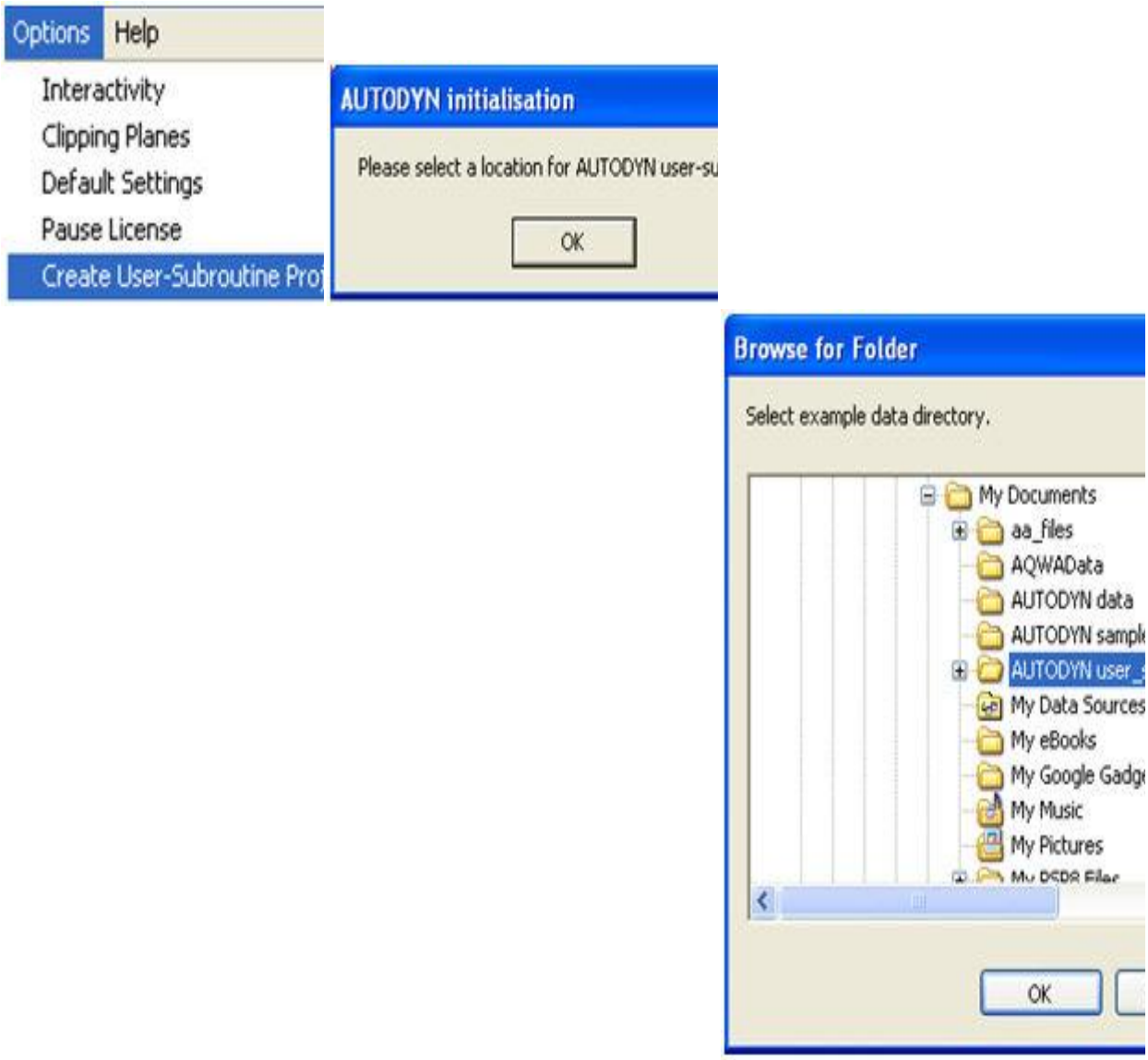
For more information on licensing please review the ANSYS, Inc. Licensing Guide.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Pull-down Menu - Options - Create User-Subroutine Project

When AUTODYN version 12 is run for the first time by the logged on user, AUTODYN will perform a few additional initialization steps to create user specific settings.

In previous version you also would have been prompted for a location for user-subroutine files which would have been created after the initial step. This step is no longer performed and the generation of the user-subroutine project is now available as a user action from within AUTODYN from the Options Menu instead.



After selection of the location for the user-subroutines AUTODYN will install the user subroutine project in a sub-directory with name *AUTODYN user_subroutines.*

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

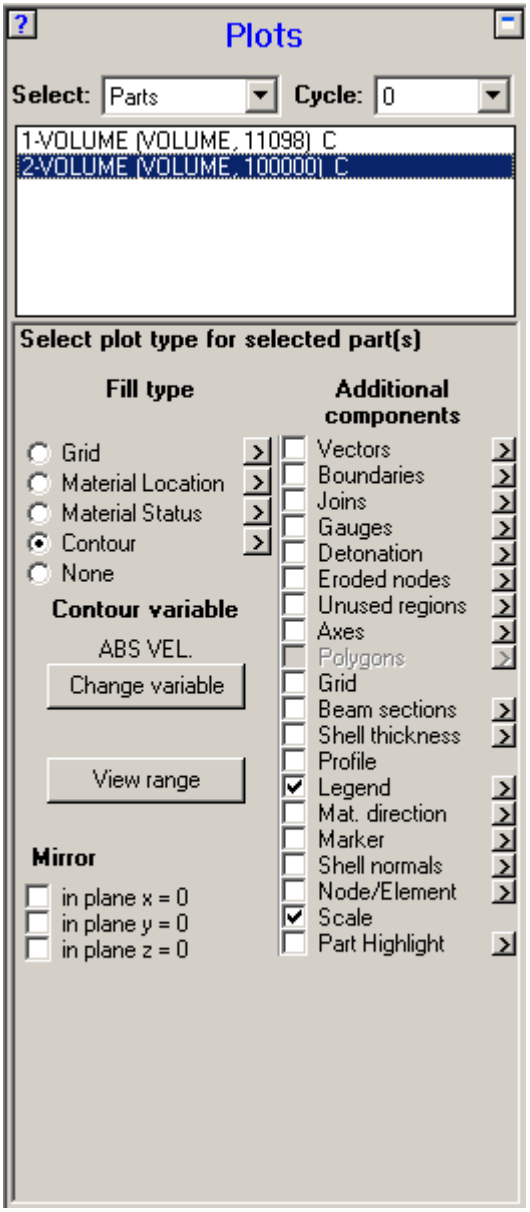
Pull-down Menu - Help

This pull-down menu provides general help for AUTODYN (Contents), and provides information about the current version (About).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Plots

Plots



This panel lets you set up what you want to see in the View panel.

Cycle

This pull-down menu lets you choose which cycle you want to view for the current model (Ident).

Select part(s)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This box lists the Parts in the model. Operations performed in the Plots panel will only be applied to the Part(s) you select

Fill type

These options let you select the basic fill type you want to view. Select only one fill type.

Additional components

This option let you view additional components in your plot. Check the boxes alongside the additional components you want to include (any number can be selected).

Contour Variable

This button is only active if a Contour Fill type is selected. Press this button to change the variable to be contoured.

View Range

This button is only active if a single Part is selected. Press this button to limit the IJK range that is plotted for the Part.

Mirror

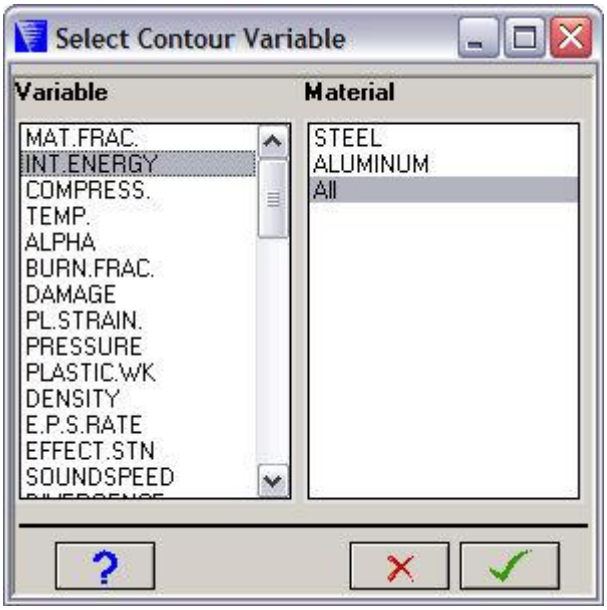
Check a box to mirror the image in the associated axis.

All Fill types and Additional components have default **Settings**. You can quickly access and change these settings by pressing the button to the right of these options. Doing so will bring up a dialog window containing the settings available for the associated option.

The **Settings** button on the Navigation bar gives you access to all these settings.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Plots - Select Contour Variable



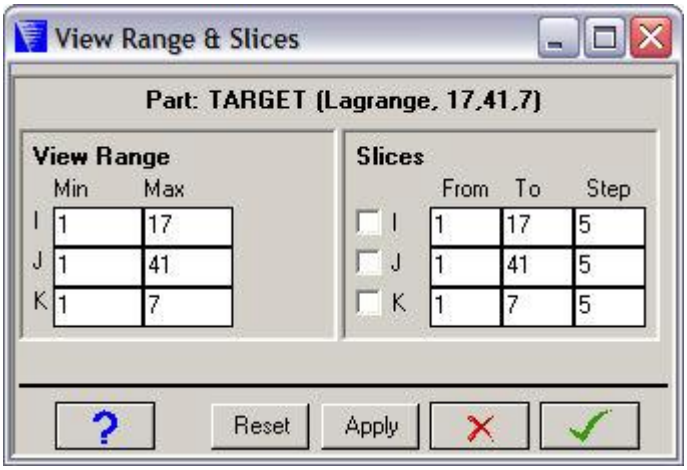
This dialog lets you select the variable you want to contour.

Select the variable from the left hand list.

For multi-material variables (those at the top of the left hand list), you must also select a material (or "All") from the right hand list.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Plots - View range and slices (Structured)



This dialog window lets you set the view range and slices to be used to display a structured part of your model.

Part:

The part that you are defining a view range and slices for is shown here, together with its index range.

View Range

This box lets you define the I, J and K range you want to see for this Part.

Slices

Instead of a solid view of your Part, you can request to see only slices (planes) through the part for any of the three indices.

Check the box next to an index for which you wish to create slices and define the planes for which you want to see these slices.

If no boxes are checked, a solid plot using the view range will be shown.

Reset

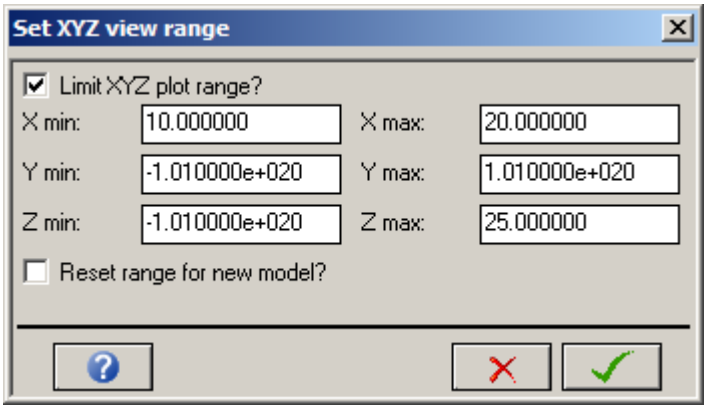
This button resets the View Range and Slices to their default values.

Apply

Pressing this button applies the current settings.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

View range and slices (SPH/Unstructured)



Limit XYZ plot range?

Select this option to truncate plotting of unstructured and SPH parts. Any unstructured elements or SPH nodes that fall outside the view range specified will be eliminated from the plot.

View Range

Specify the maximum and minimum X, Y and Z coordinates to be plotted for unstructured and SPH parts. These limits are not set on a per part basis, and so setting limits will truncate the plotting for all unstructured and SPH parts in the model.

Reset range for new model?

If this option is selected, when a new model is opened, the view range will be reset to show the full model. If this option is not selected, the view range you have selected will be maintained.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

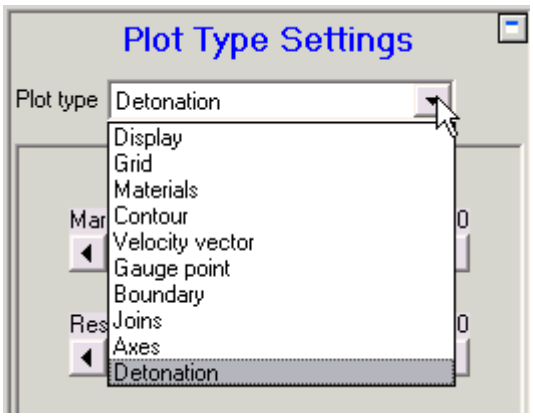
Settings

Settings

Selecting **Settings** on the Navigation Bar displays the **Plot Type Settings** panel.

This panel lets you change settings for the various plot types that control how you view your model.

At the top of this panel, a pull-down menu lets you to select the plot type whose settings you wish to change :



You can change settings for the following plot types:

Display
Grid
Materials
Contour
Velocity Vector
Gauge Point
Boundary
Joins
Axes
Detonation

These settings can also be changed by pressing next to the option in the **Plots panel.**

The following additional settings can also be changed by pressing next to the option under **Additional Components** in the **Plots panel.**

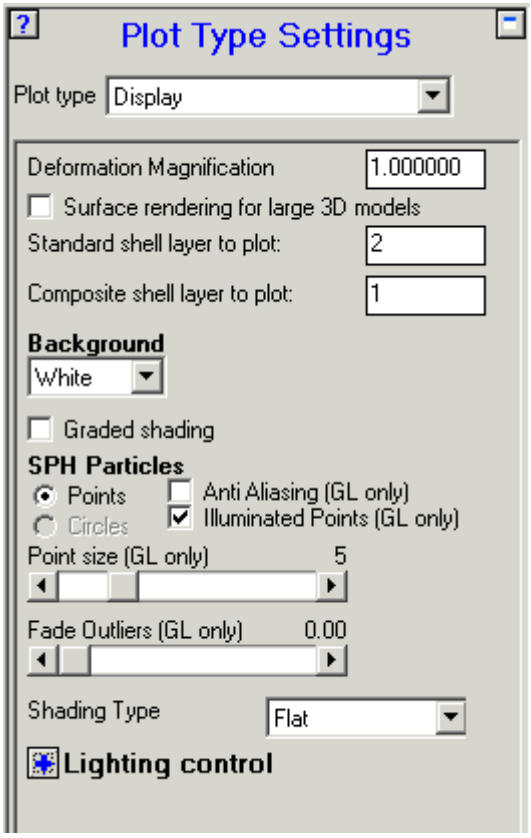
Eroded Nodes
Unused Regions
Polygons (2D only)
Beam sections
Shell thickness

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Legend
Profile
Mat. direction
Marker
Shell normals
Node/Element
Part Highlight

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Display



This panel lets you set the following general options :

Deformation Magnification :

This option lets you view magnified grid deformations. Input the magnification you want to use (1.0 gives no magnification).

Surface rendering of large 3D models :

3D models using hexahedral elements (e.g. Lagrange or Euler) are usually rendered using all the elements. Check this box to render models using only external faces. This makes the process much quicker. This option cannot be used for all plots (e.g. plots with opacity applied).

Standard shell layer to plot :

This option lets you set the layer that is plotted for standard shells (e.g. in contour plots, material status plots etc.)

Composite shell layer to plot :

This option lets you set the layer that is plotted for composite shells (e.g. in contour plots, material status plots etc.)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Background

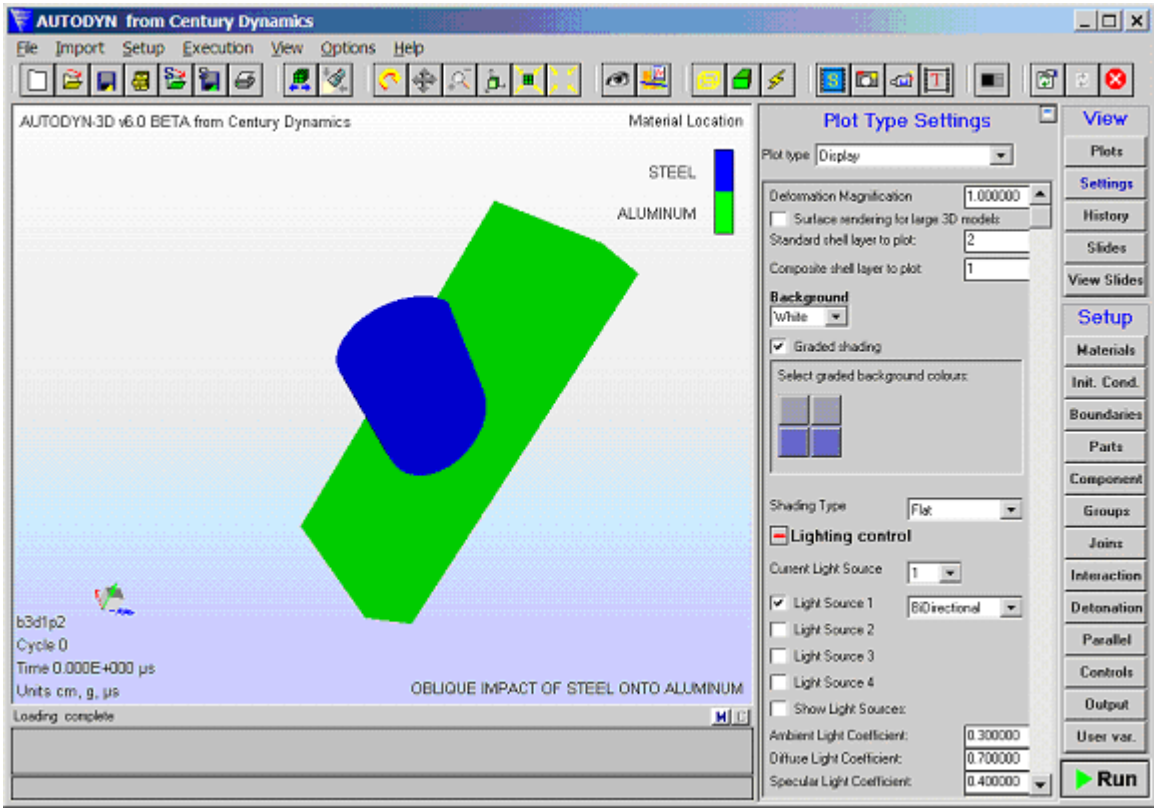
This option lets you set the background color for the view panel.

Graded Shading

This option lets you control the color, including color grading, for the background of the View panel.

You may select a color to be used for each of the four corners of the display by pressing the buttons

The selected (Default) colors shown here will produce the following background

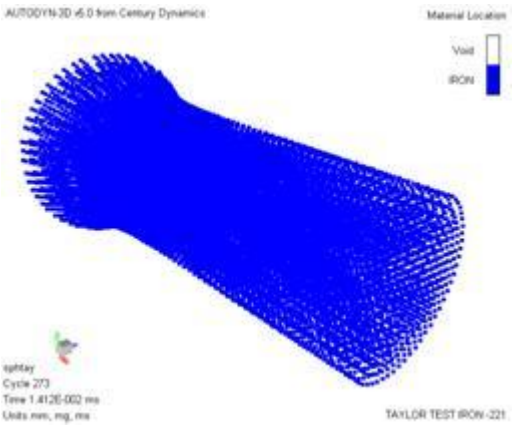


SPH Particles

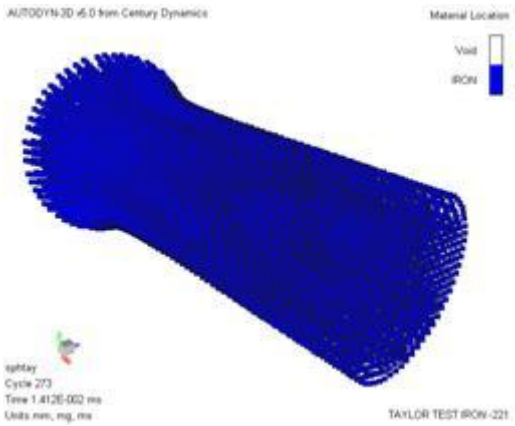
Controls for the display of SPH particles are only shown when the model contains SPH parts. In this section, the controls for displaying the points with anti-aliasing and illuminated points are accessible. In addition, control of particle display size and amount of fade for outliers can be set.

The figures below compare plots with and without the illuminated points option active.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



SPH representation without Illuminated Points

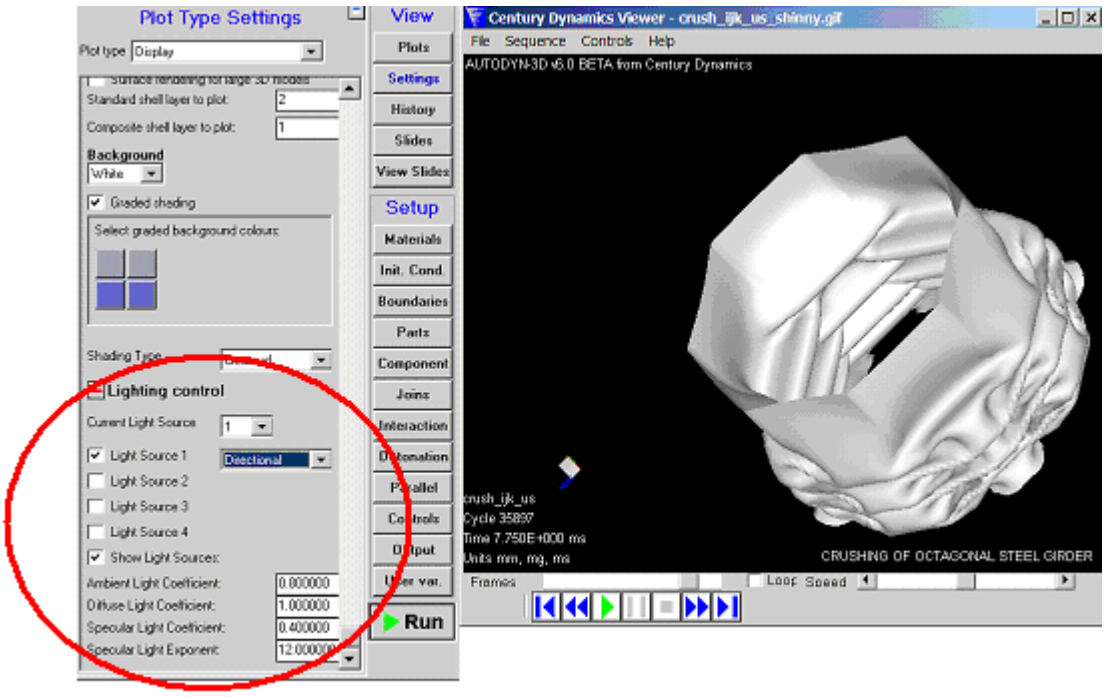


SPH representation with Illuminated Points

The fade outliers slider gives you the ability to fade in and out of view the sph particles which exist outside of the range set in the contour panel (when using the truncate facility), this facility is only available when illuminated points are displayed.

Advanced Image Shading and Lighting Options

As set of advanced shading and lighting options are available



Using a combination of the above options, metallic looking surfaces can be rendered, for example

Shading type

The shading type can now be selected as either flat or Gouraud. For flat shading every point on a face is given the same intensity. For Gouraud shading, the mesh is

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

shaded so as to give the appearance of curved surfaces. The intensity for each point is interpolated from the normal of the face vertices.

Flat shading

Gouraud shading

Lighting Control

The advanced lighting control lets you specify up to 4 different light sources and also to control the direction and form of each light source.

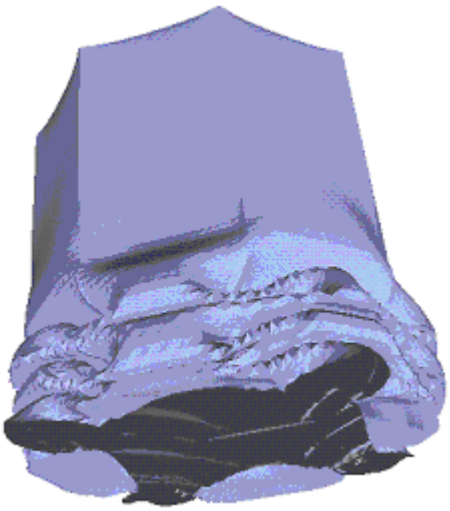
The following options are available:

Current light source
This allows you to select the current light source to be interactively transformed, after selecting the "Transform Light" button on the toolbar.

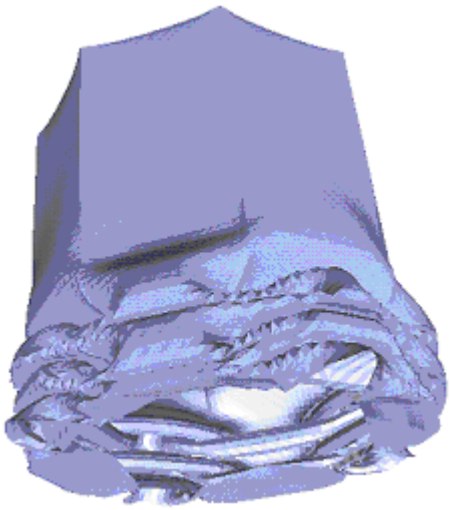
Light source toggles
You can choose the number of light sources required (up to a maximum of 4).

Light source type
You can select between directional, bi-directional, spot or point light sources. Examples of these are shown in the figure below.

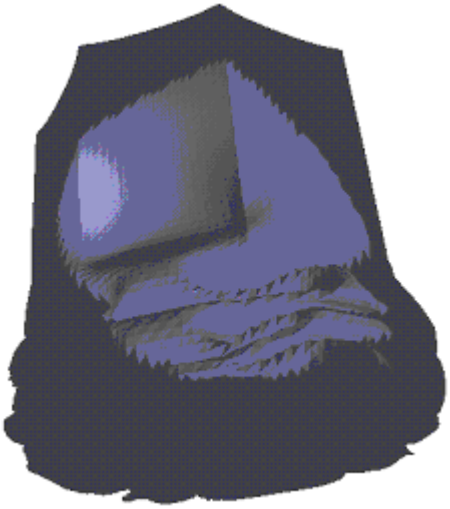
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



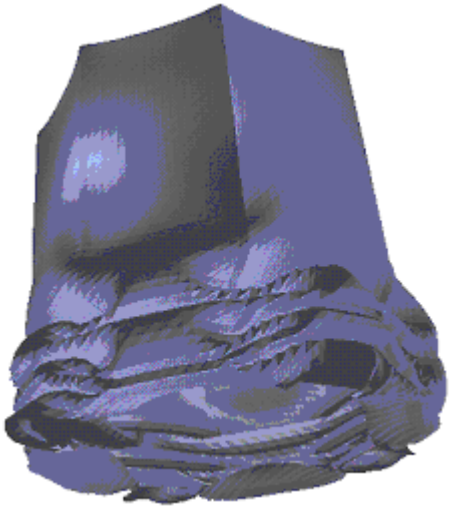
Directional Light Source



Bi-Directional Light Source



Spot Light Source



Point Light Source

Show light sources

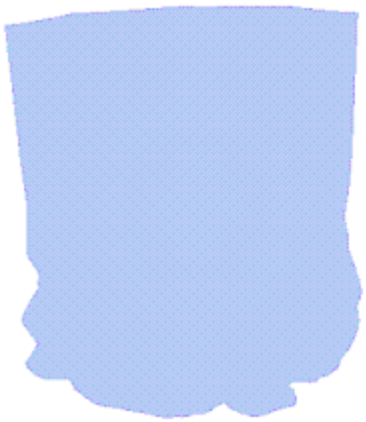
The direction and position of the light sources is shown on screen by white arrows (switch the background color to black). Examples are shown in the figure below (from left to right) for bi-directional light source, spot light source, point light source and directional light.

Ambient, Diffusive and Specular Light Coefficients

The ambient light coefficient controls the amount of background lighting. Ambient light is non-directional, affecting all parts of all surfaces equally.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

No lighting



Ambient lighting only

Diffuse light reflectance controls the proportion of the available non-ambient light that the object reflects equally in all directions. Non-ambient light emanates from the directional and point light sources.

Specular highlights of a particular color and brightness are created when the direction of incoming light (from a directional or point light source) is "sufficiently close" to the viewing direction. The specular reflection coefficient determines the brightness of such highlights.

Diffusive light only

Specular light only

The specular exponent determines what "sufficiently close" means. The greater the sharpness, the smaller (more focused) the size of the specular highlight.

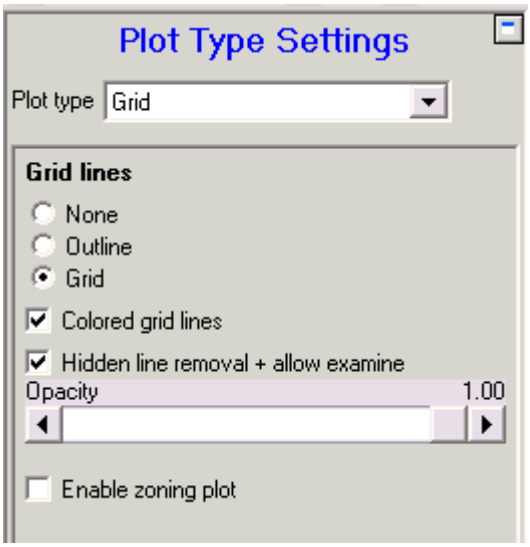
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

High specular exponent

Low specular exponent

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Grid



This panel lets you set the following options for grid plots (some options are not present for 2D models) :

Grid lines :

These options let you view grids with or without grid lines (or with just the outline of the grids displayed).

Colored grid lines :

Check this box to display grid lines in color (otherwise they are displayed in black).

Hidden line removal + allow examine :

Check this box to view grids with hidden line removal. The Examine feature is only available if hidden line removal is chosen

Opacity :

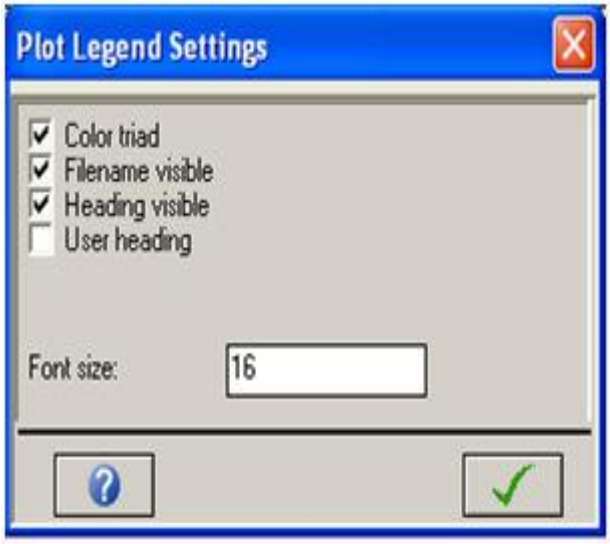
This slider lets you set the opacity for your view. It is only functional if hidden line removal is selected. An opacity of 1.0 means you cannot see inside a grid. An opacity of 0.0 is equivalent to turning hidden line removal off (i.e. you can see everything inside the grid).

Enable zoning plot :

By default you will only see the parts of a grid that are filled with material. Check this box to see all zoning, including unused elements.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Legend



This window lets you set the plot legends you want to display in the View panel.

Color triad

Check this box to show a colored triad in the lower left corner of the View panel (otherwise the triad is monochrome).

The axes are color-coded as follows :

Red	X
Green	Y
Blue	Z

Filename visible

Check this box if you want the filename (ident) of you model to be visible in the bottom left corner of the View panel.

Heading visible

Check this box if you want the heading of your model to be visible in the bottom right corner of the View panel.

User heading

Check this box if you want to override the heading of you model with an alternative (user) heading. Enter the user heading in the field provided.

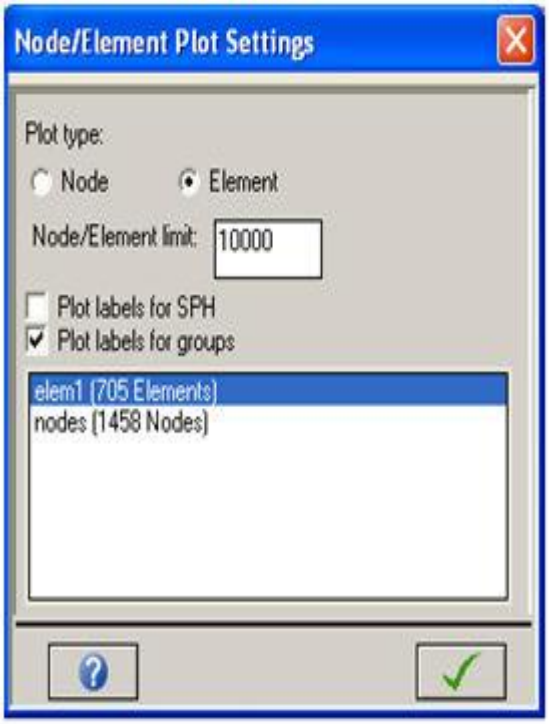
Font size

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the size of the font used in the graphics window for the legend, scale and user heading.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Node/Element Numbers



This window lets you change settings for plotting node/element numbers

Plot type
Choose to plot numbers for nodes or elements.

Node/Element limit
The maximum number of node or element numbers to be plotted. For larger models this number may be increased which can result in a slower graphic refresh.

Plot labels for group
Select this option to plot node or element numbers for a particular group. If this option is not selected, node/element numbers are shown for all plotted parts.

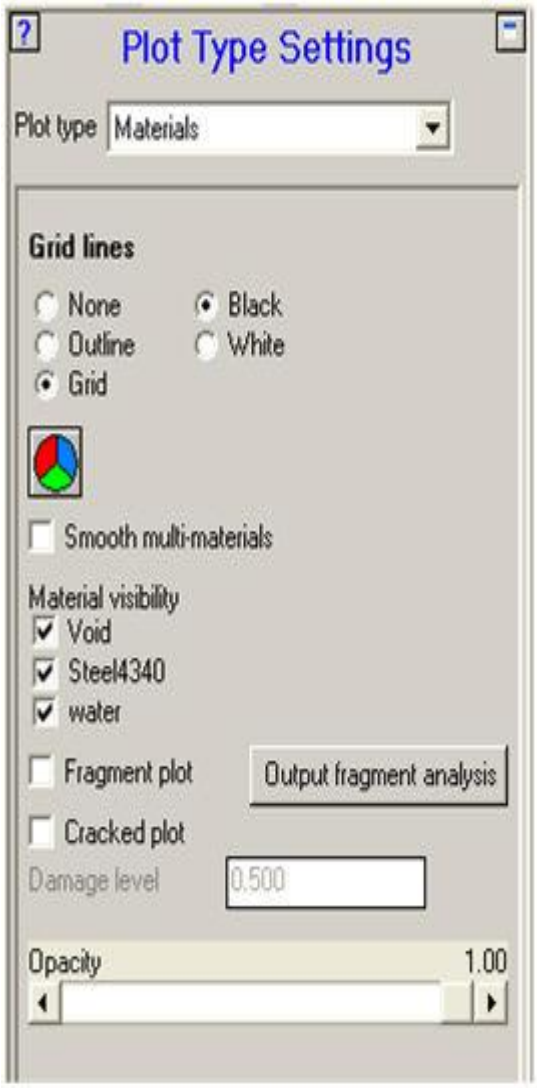
Group list
If the "Plot labels for group"

Plot labels for SPH
Select this option to include SPH particles in the node or element plot.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Materials

Settings - Materials



This panel lets you set the following options for material plots:

Grid lines :

These options let you view grids with or without grid lines (or with just the outline of the grids displayed) and the color in which the grid lines or outline are displayed.

Pressing this icon lets you set colors for each material using the **Modify Material Color** dialog window.

Smooth multi-materials

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box to show smooth multi-material interfaces

Material visibility

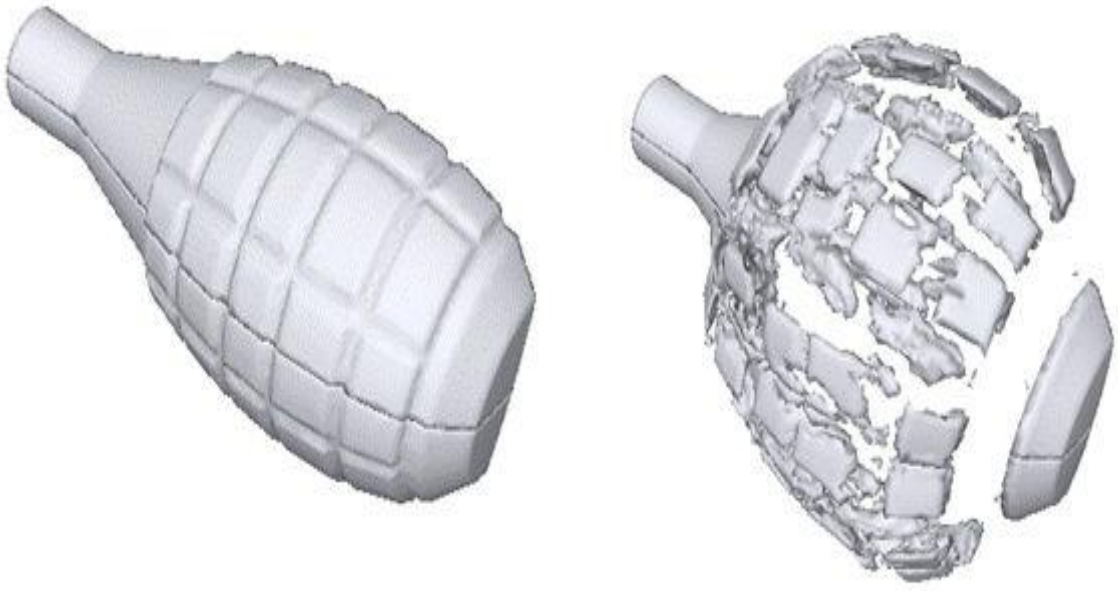
Check the boxes for materials you want to be visible in the view.

Fragment plot

Check this box to view a fragment plot. Fragment plots only show elements that contain material that has not failed.

Pressing the **Output fragment analysis** button causes fragment data to be output to a file.

The fragment output post processing option is now available for all solvers, including the unstructured solvers.



Simulation of fragmentation of a grenade modeled using unstructured tet elements

Cracked plot

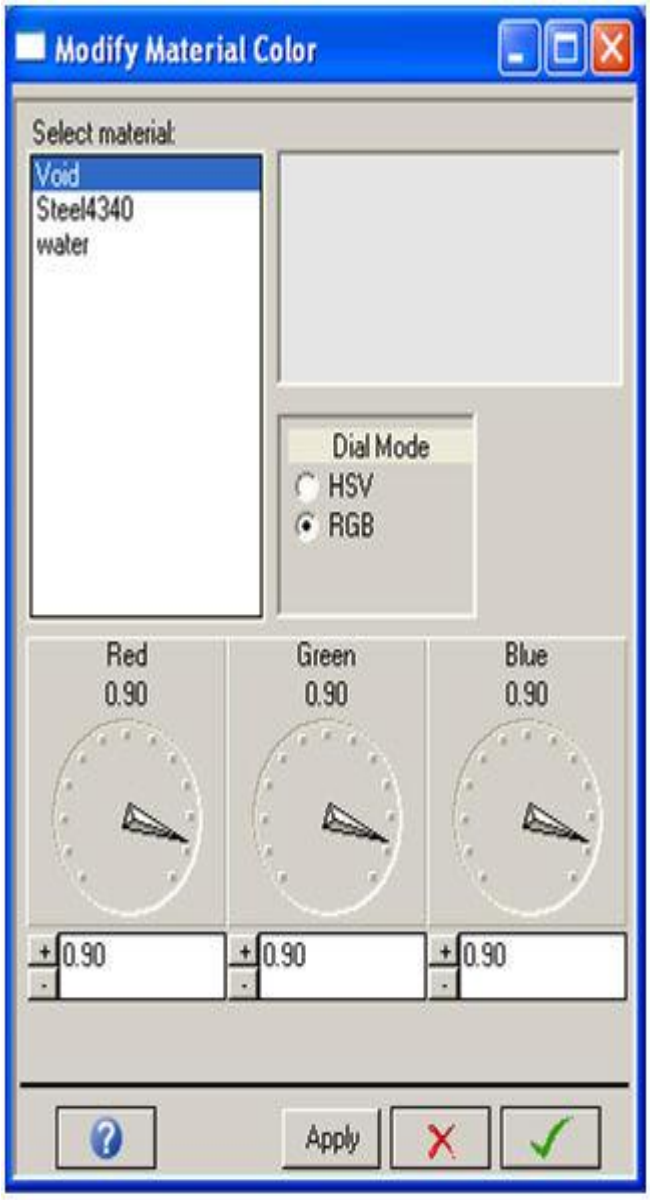
Check this box to view a cracked material plot. A cracked material plot shows all elements that contain material with damage above the specified **Damage level.**

Opacity

This slider lets you set the opacity for your view. An opacity of 1.0 means you cannot see inside the grid. An opacity of 0.0 is equivalent to turning hidden line removal off (i.e. you can see everything inside the grid).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Materials - Modify Material Color



This dialog lets you modify a material color.

1) Select the material whose color you wish set from the list (top left).

2) Choose the dial mode you wish to use to set the color.
- HSV - Hue, Saturation, Value
- RGB - Red, Green, Blue

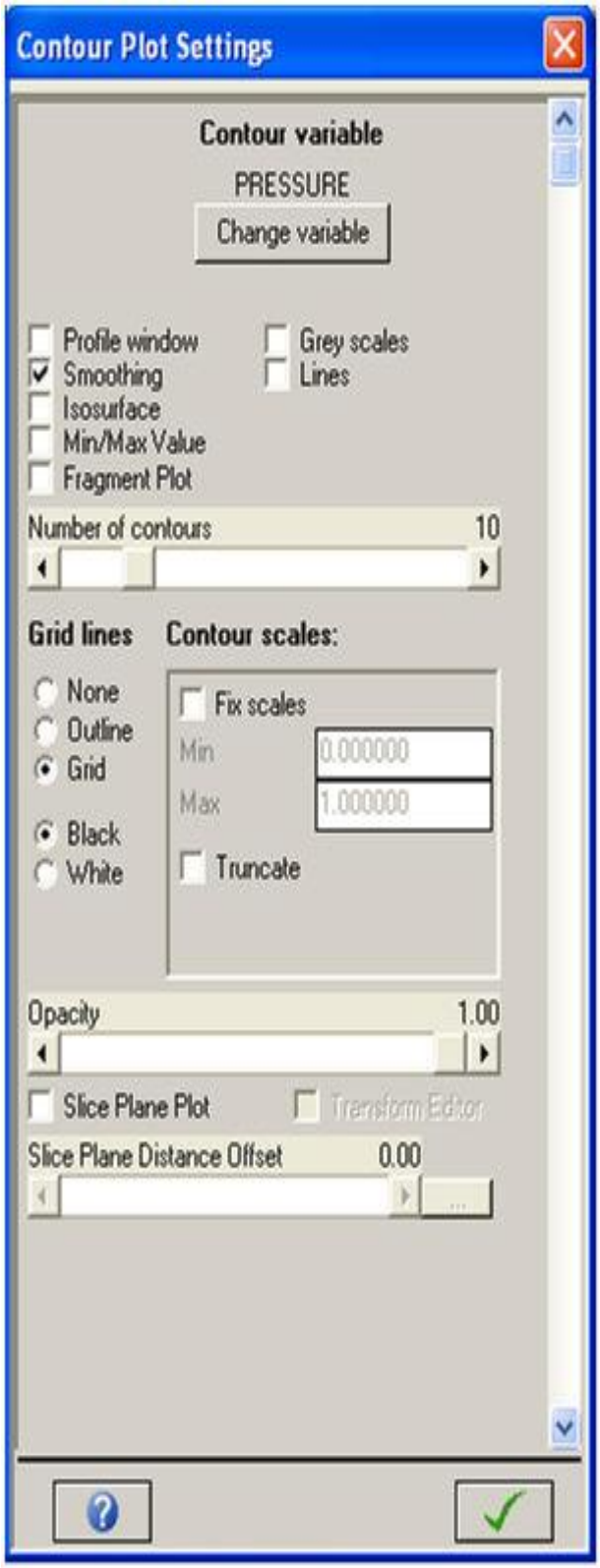
3) Use the three dials at the bottom of the window to set the color.

As you turn the dials, the top right window displays the color that will be used.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Contour

Settings - Contour



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This panel lets you set the following options for contour plots (some options are not present for 2D models:

Contour variable

Press this button to open the **Select Contour Variable** dialog window, allowing you to choose the variable you want to contour.

Profile window

Check this box to open the **Profile Plot** dialog window, allowing you to compose and display a profile plot.

Smoothing

Check this box if you want smoothed contour plots. All elements have a single centered value for each variable. If you do not smooth a contour plot each element will be color-coded for the value the variable has in that element. If you smooth a contour plot, then the centered values are interpolated in space to provide smoothly varying values.

Isosurface

Check this box to view a 3D isosurface plot of the contour variable. You are prompted for the value to be used for the plot.

Min/Max Value

Check this box to open a window that displays the minimum and maximum values for the contoured variable.

Grey scales

Check this box to view contour plots using grey scales instead of color.

Lines

Check this box to view contour plots with colored lines instead of colored fills.

Number of contours

This slider lets you select the number of contours you want to use for your plot.

Grid lines

These options let you view grids with or without grid lines (or with just the outline of the grids displayed) and the color in which the gridlines or outline are displayed.

Contour scales

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Usually you should let the maximum and minimum values between which contours are plotted be set automatically using the data in the model. However, you may override these settings in the **Contour scales** box. For example, if a maximum value is entered which is lower than the highest value present in the model, a red area will appear corresponding to all values above the maximum that has been set. Selecting Fix scales lets you fix the current maximum and minimum values. This is particularly useful when you want to generate a slideshow from multiple cycles of a single model. The red area may be removed by selecting **Truncate**.

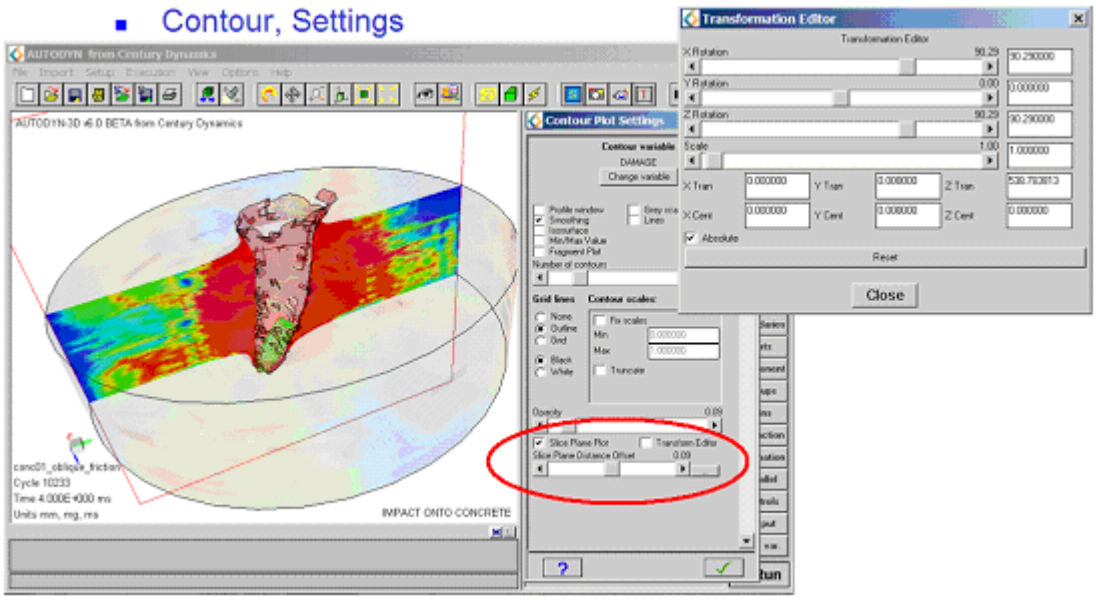
Opacity :

This slider lets you set the opacity for your view. An opacity of 1.0 means you cannot see inside the grid. An opacity of 0.0 is equivalent to turning hidden line removal off (i.e. you can see everything inside the grid).

Slice Plane Plot

The "Slice Plane" feature is a visual tool to take a section through a model at an arbitrary angle. Results values, as selected under the contour option, are contoured on the selected slice through the model.

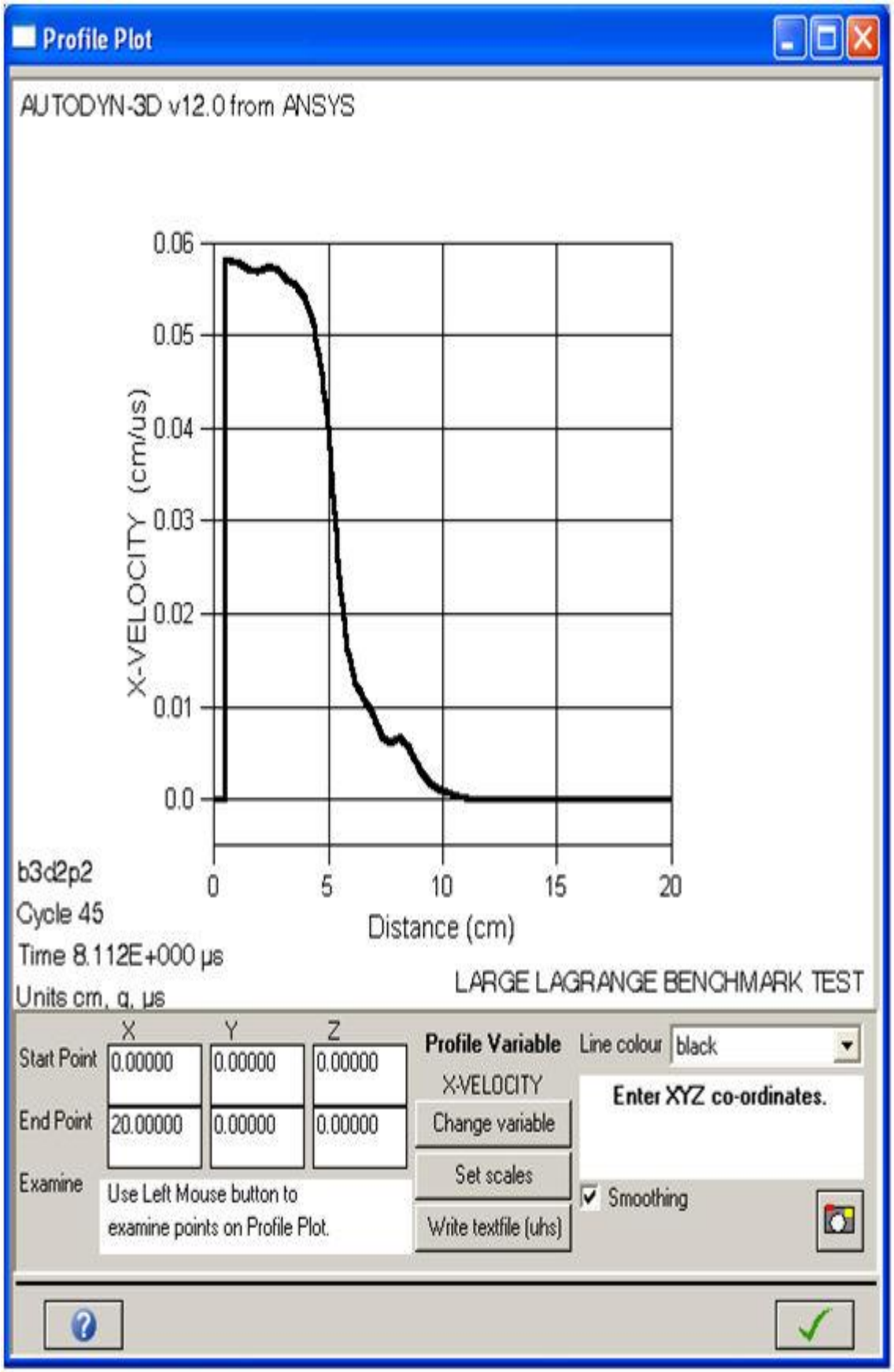
We recommend that you use the slice box feature in conjunction with the Opacity feature of the Contour plot to obtain best results.



Example slice plane plot showing the damage inside a concrete target

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Contour - Profile Plot



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This dialog window lets you compose and display a profile plot of a chosen variable along some defined line.

Define the Line along which the variable is profiled

Specify the end points for the line along which the variable is profiled.

You can do this by specifying the coordinates for the start point and end point of the line (lower left)

Alternatively, in 2D, you can select this line interactively in the View Panel using the mouse (hold shift and use the right mouse button to drag a line across your model).

Change Variable

Press this button to change the variable you want to profile.

Smoothing

Check this box if you want to see a smoothed profile plot.

Line Color

Use this pull-down menu to select the line color for your profile plot

Write textfile (uhs)

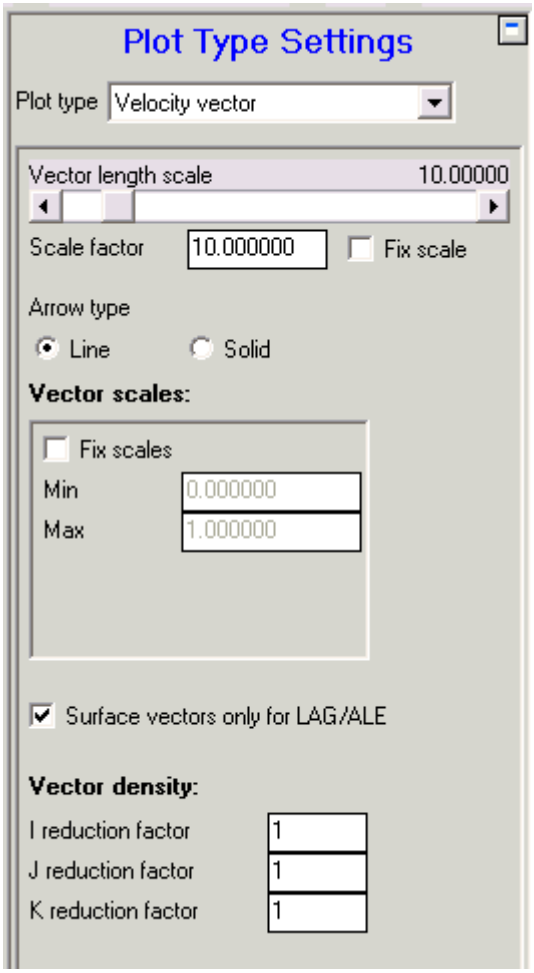
Press this button to write digital history data for the current plot to a text file. The text file has an extension **.uhs** and is Microsoft Excel compatible.

Examine

Use the Left Mouse Button to select a point on the graph. The corresponding x and y values are displayed in the Examine panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Velocity vector



This panel lets you set the following options for velocity vector plots:

Vector Length Scale

This slider lets you set the scale factor of the arrows that are drawn for this plot type. You can alternatively type in a scale factor in the box below the slider. You can also fix this scale factor so that all plots use the same scale. The range for this factor is calculated from the maximum velocity and the physical dimensions of your model.

Arrow Type

This option lets you choose the type of arrows that are plotted. Solid arrows take longer to plot than line arrows (the default).

Vector scales

AUTODYN chooses vector scales automatically using the maximum and minimum velocities in the model. You can override these scales and set your own minimum

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

and maximum scale values in this box. You can also fix these scales for all plots (useful when creating slideshows).

Surface vectors only

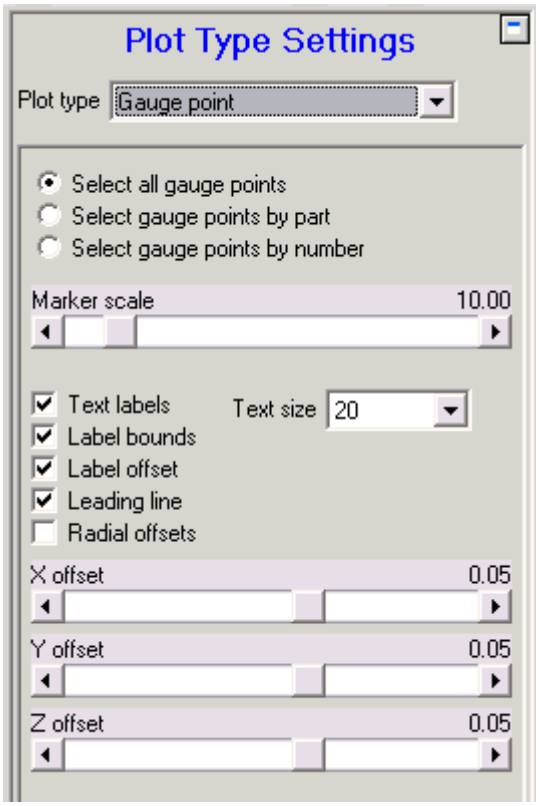
Check this box if you only want to plot vectors on material surfaces. This can speed up plotting significantly, but is not always what you need, particularly if your plot uses opacity.

Vector density

This option lets you reduce the number of velocity vectors plotted, which can speed up plotting significantly. The option only applies to Lagrange and ALE parts. You specify the reduction factor for each grid index (I,J and K).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Gauge point



This panel lets you set the following options for gauge point plots:

Select gauge points

These options let you select all gauge points, all gauge points associated with a part or individually by number. For the last two cases additional dialog prompts you for the part names or gauge numbers as required.

Marker scale

Each gauge point is represented by a marker. This slider lets you set the marker size.

Text labels

Check this box to label each marker with its corresponding gauge number.

Text size

This defines the text size used for the label

Label bounds

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box to put a bounding box around each label.

Label offset

Check this box to set your own offset position for labels

Leading line

Check this box if you want to show lines joining each marker to its label

Radial offset

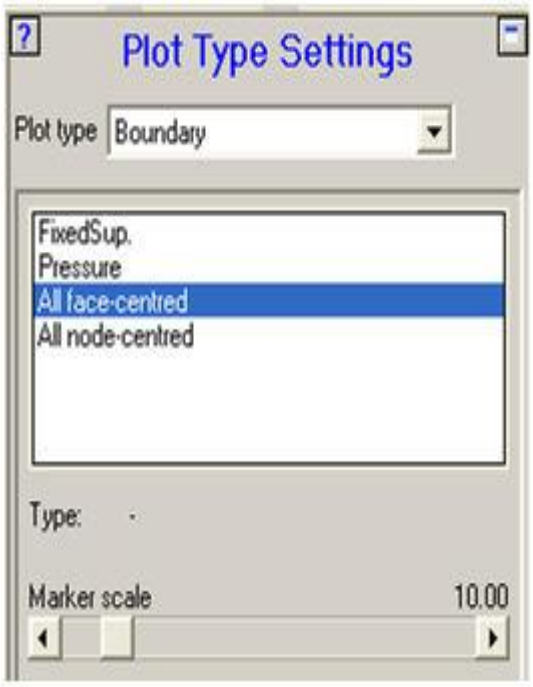
Check this box if you want to use radial label offsets instead of linear offsets

X / Y / Z offset

These sliders let you adjust the label offset distance in the X, Y and Z directions.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Boundary



This panel lets you set the following options for viewing boundary conditions :

Boundary Condition

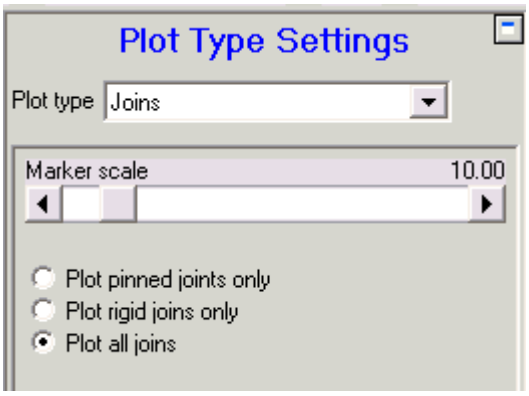
This pull-down menu lets you choose the boundary condition you want to view.

Marker scale

This slider lets you choose the size of the marker used to show where nodal boundary conditions have been applied.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Joins



This panel lets you set the following options for viewing boundary conditions :

Marker scale

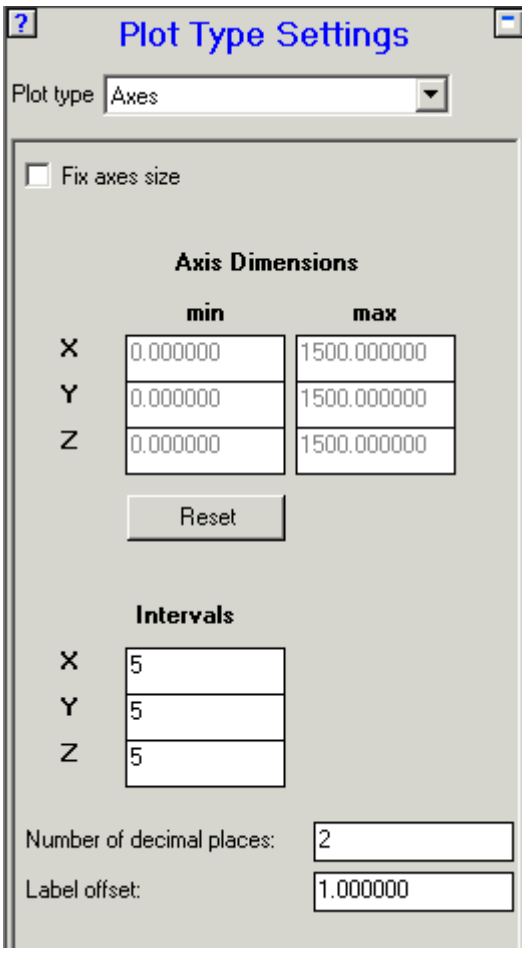
This slider lets you choose the size of the marker used to show joined nodes.

Joins to plot

These options let you choose the type of joins you wish to plot (pinned, rigid or all joins)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Axes



Axes are automatically chosen to fit the minimum / maximum range of your model.

This dialog lets you define your own axes.

Fix axes size

Check this box to define your own axes.

Axis Dimensions

Set the min / max range for each of your axes.

Reset

This button resets the min / max ranges to those automatically set from your model dimensions.

Intervals

Set the number of intervals you want displayed on each of your axes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

with your own. It is possible for the user to fix the size of the axes in the axes settings panel.
To reset the axes size to the default, use the Reset button. The number of intervals on the axis can also be controlled.

Number of decimal places

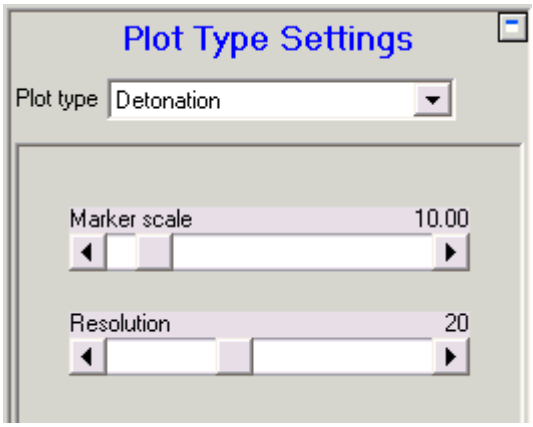
Enter the number of decimal places to be displayed for the axes.

Label offset

Set the label offset to control the distance between the axes and the labels.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Detonation



This panel lets you set the following options for viewing detonation points :

Marker scale

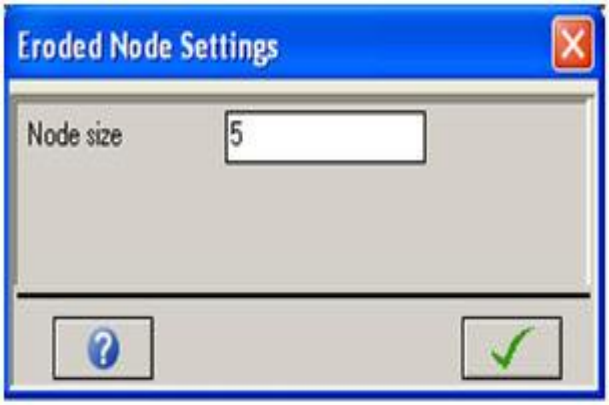
This slider lets you choose the size of the marker used to show detonation points.

Resolution

This slider lets you choose the marker resolution.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Eroded Nodes



This window lets you set the node size for plotting eroded nodes.

Enter an integer number. The higher the number, the larger the node.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Unused Regions

This window lets you set options for the plotting of unused regions.

Gridlines

Select whether you want to plot the outline or grid for unused zones and whether the outline/grid is drawn in black or white.

Plot contours

Check this box if you want the contour variable contoured on unused faces (for contour plots). Otherwise the face is displayed in grey.

Opacity :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This slider lets you set the opacity for unused regions. An opacity of 1.0 means you cannot see inside the region. An opacity of 0.0 is equivalent to turning hidden line removal off (i.e. you can see everything inside the region).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

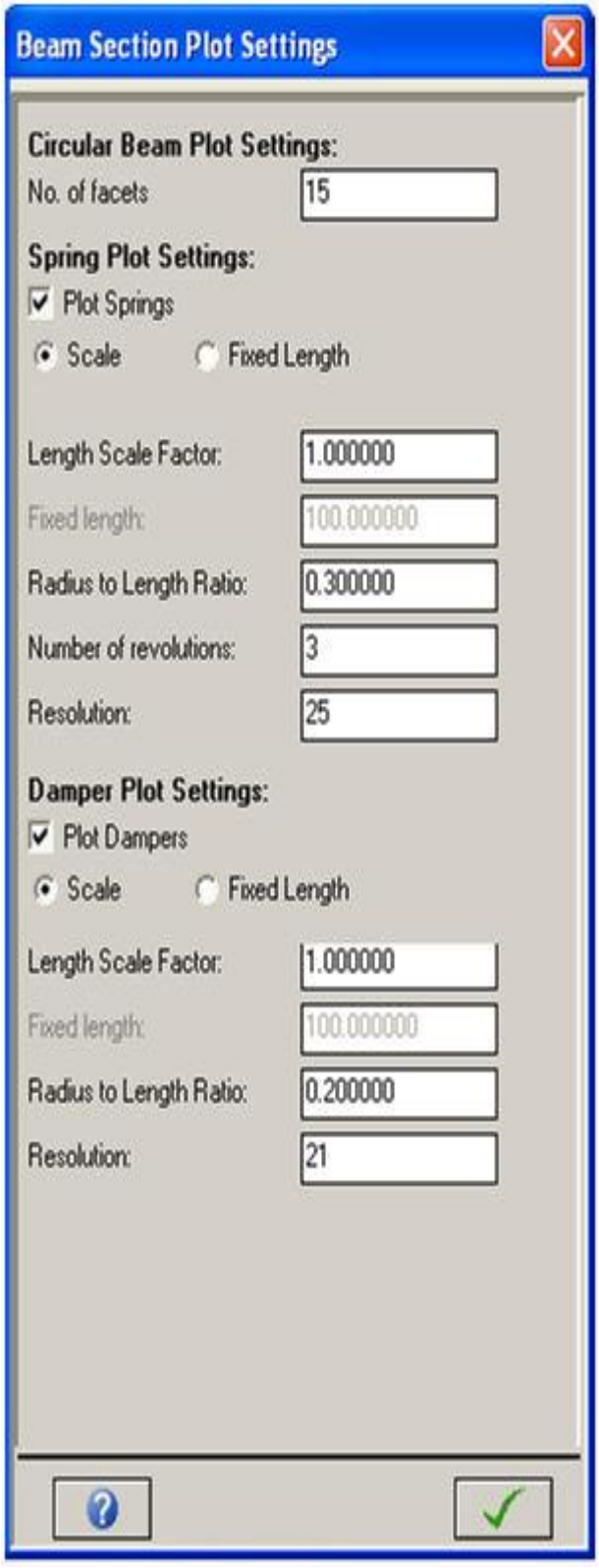
Settings - Polygons

This window lets you turn on the plotting of polygons (used in 2D Euler-Lagrange coupling).

Check the box to turn outline plotting on.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Beam Sections

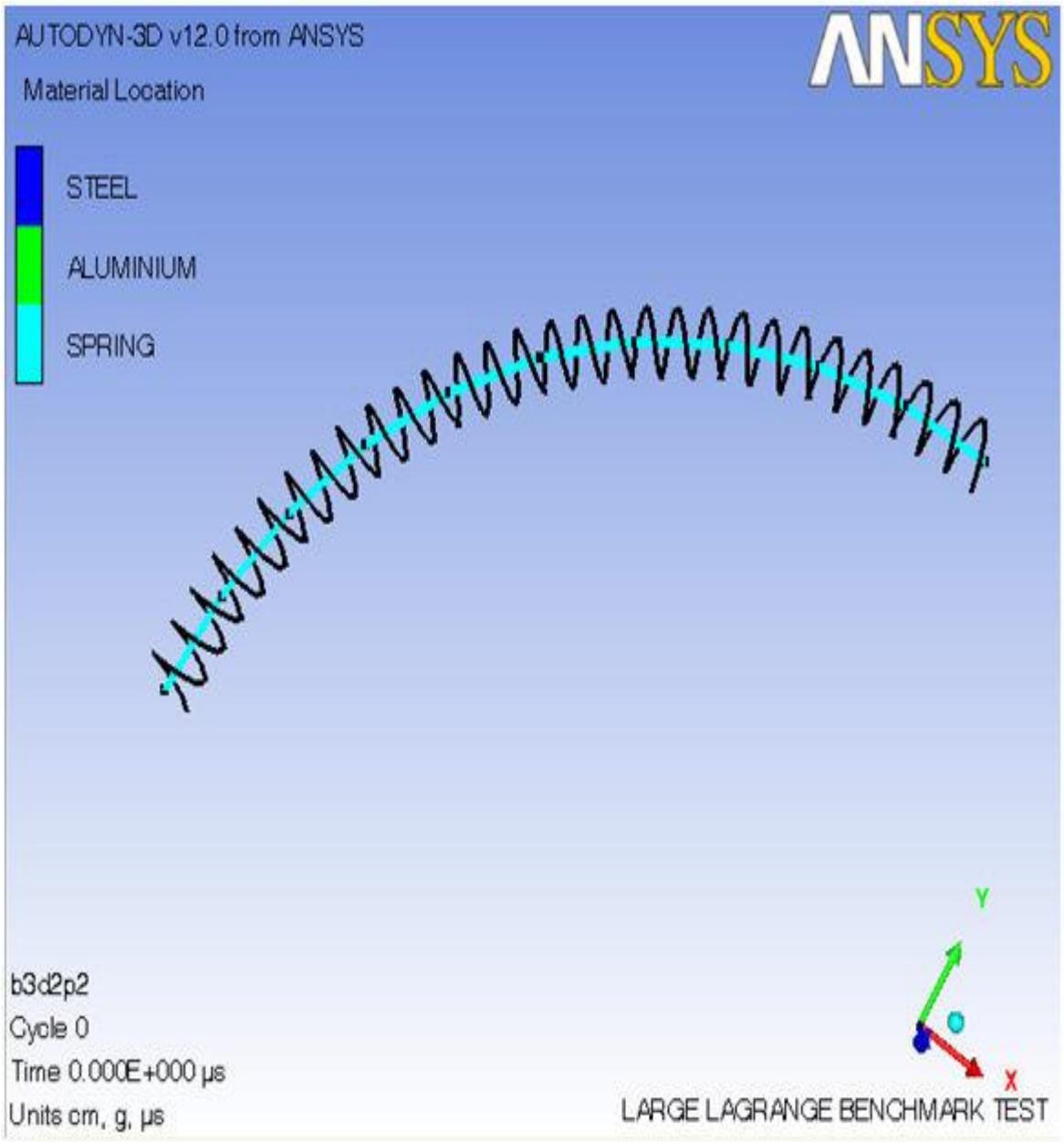


This window lets you control the plotting of beams.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Circular Beam Plot Settings
Select the number of facets to be plotted for each circular cross-section beam.

Spring Plot Settings
Select the Plot Springs toggle to enable the spring plot.



Scale
Select this option to normalize each plotted spring with the length of the spring element.

Fixed Length
Select this option to display each plotted spring as a fixed size.

Length Scale Factor

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The Length Scale Factor is only used together with the scale option. If a length scale factor of one is chosen, each spring will be displayed as the same length as the element.

Fixed Length
If the fixed length option has been selected, specify the length of each plotted spring in this field.

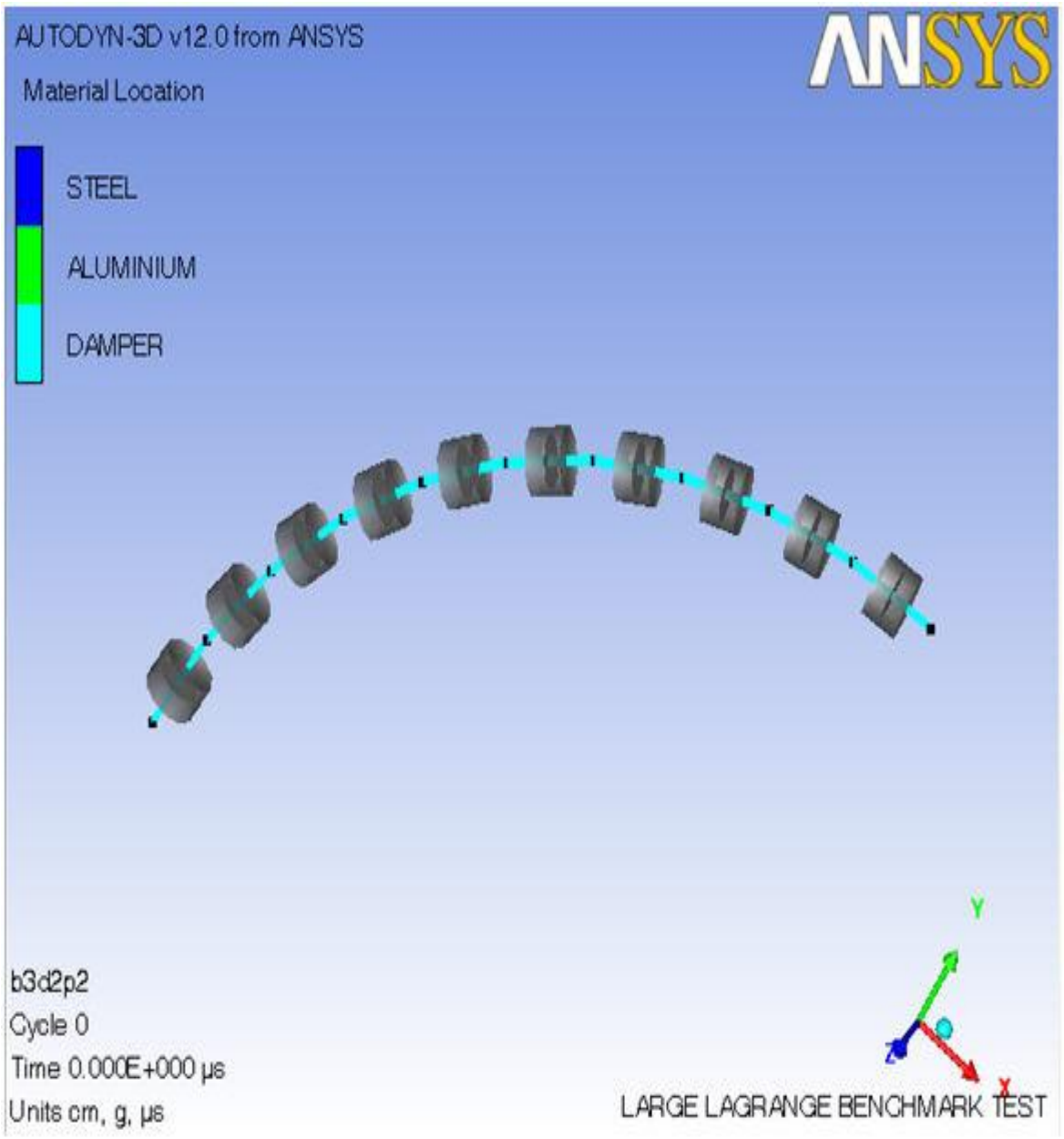
Radius to Length Ratio
Set the radius to length ratio of each plotted spring in this field.

Number of revolutions
Set the number of revolutions for each plotted spring in this field.

Resolution
Set the number of lines used to represent each plotted spring in this field.

Damper Plot Settings
Select the Plot Damper toggle to enable the damper plot.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Scale
Select this option to normalize each plotted damper with the length of the damper element.

Fixed Length
Select this option to display each plotted damper as a fixed size.

Length Scale Factor
The Length Scale Factor is only used together with the scale option. If a length scale factor of one is chosen, each damper will be displayed as half the length as the element.

Fixed Length
If the fixed length option has been selected, specify the length of each plotted damper in this field.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Radius to Length Ratio
Set the radius to length ratio of each plotted damper in this field.

Resolution
Set the number of lines used to represent each plotted damper in this field.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

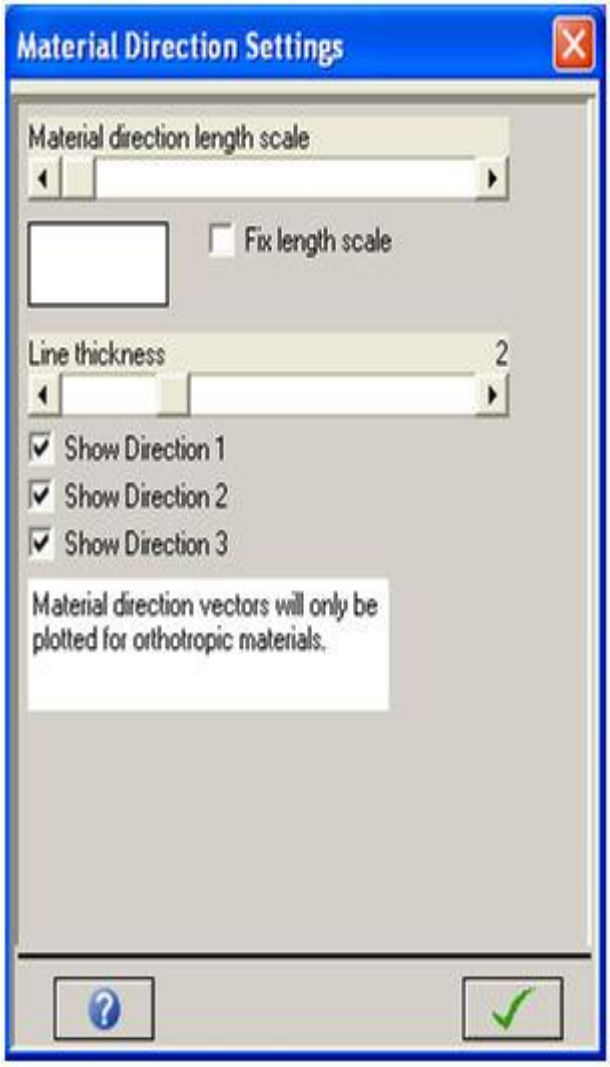
Settings - Shell Thickness

This window lets you select the thickness you want to use for rendering shells.

You can use either the true shell thickness or the artificial thickness that is used for coupling shells to Euler.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Material Direction



This window lets you change settings for plotting material directions.

Material direction length scale

Use this slider to vary the length of the lines displaying the local axes.

Fix length scale

Check this box to fix the local axes length scale.

Line thickness

Use this slider to vary the thickness of the lines displaying the local axes.

Show Direction 1

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box to display the local axis for direction 1.

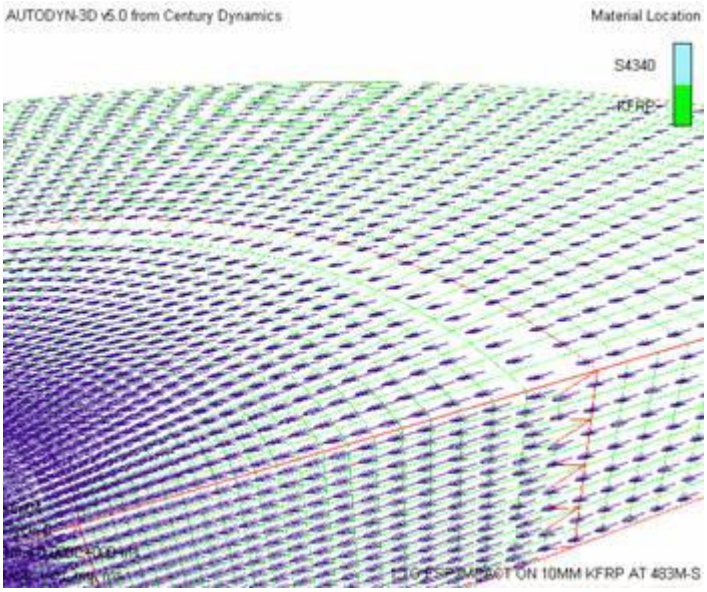
Show Direction 2

Check this box to display the local axis for direction 2.

Show Direction 3

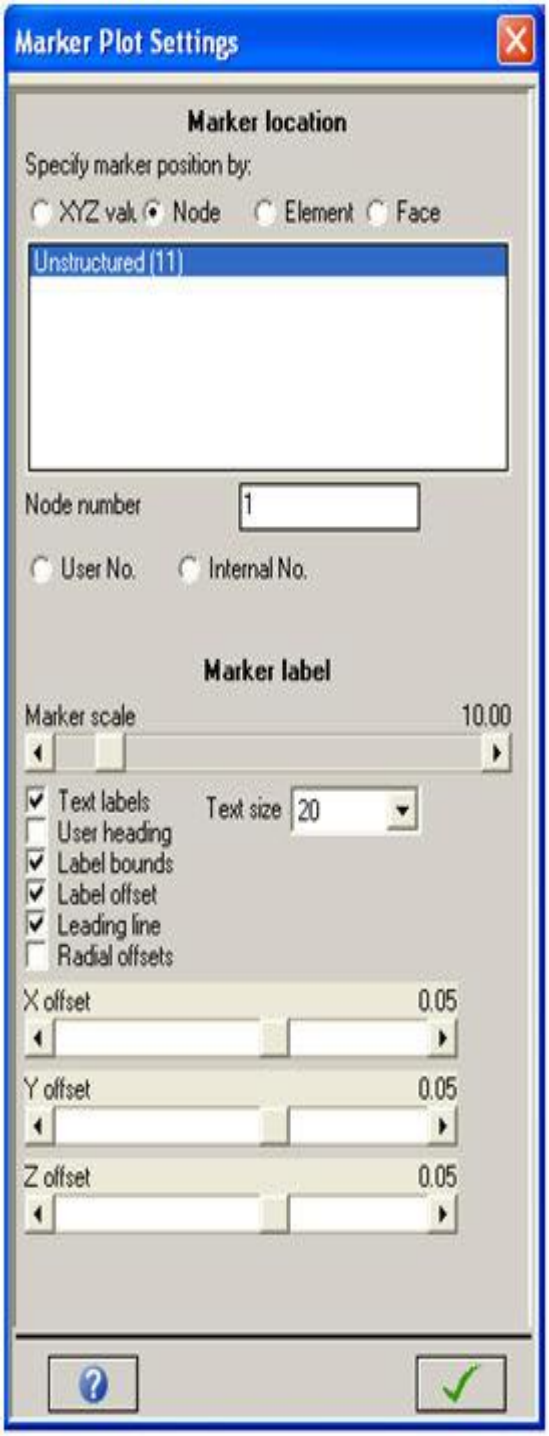
Check this box to display the local axis for direction 3

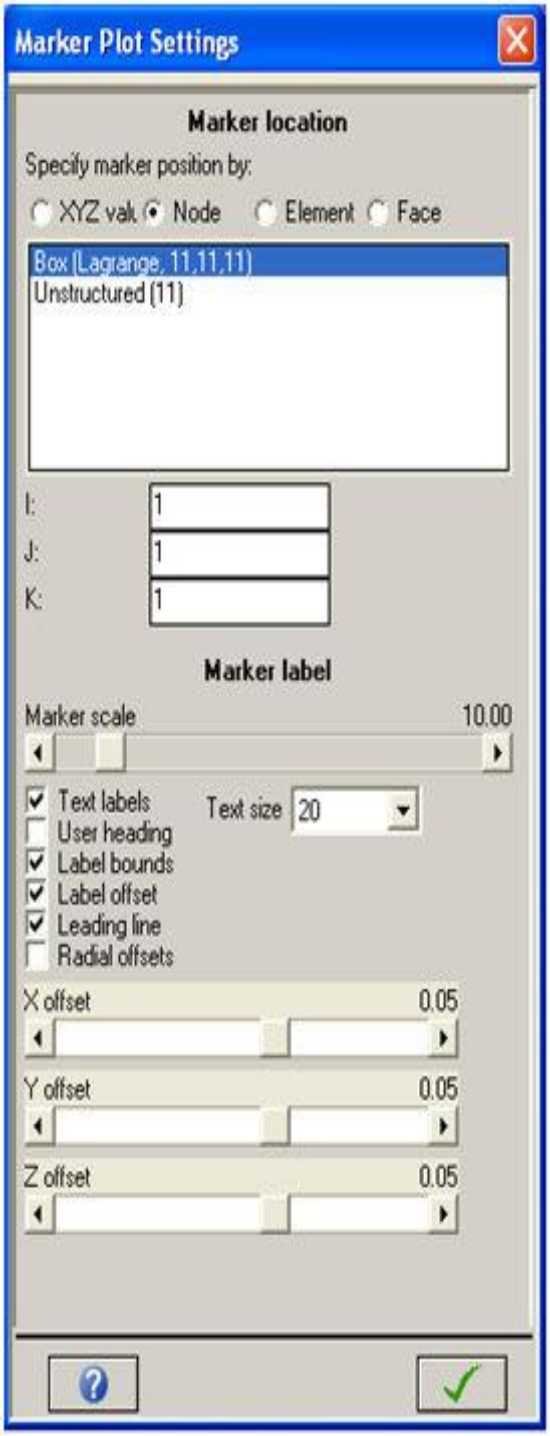
For large models containing mainly volume elements, you will get better results by selecting the Settings, Display, Surface rendering option.



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

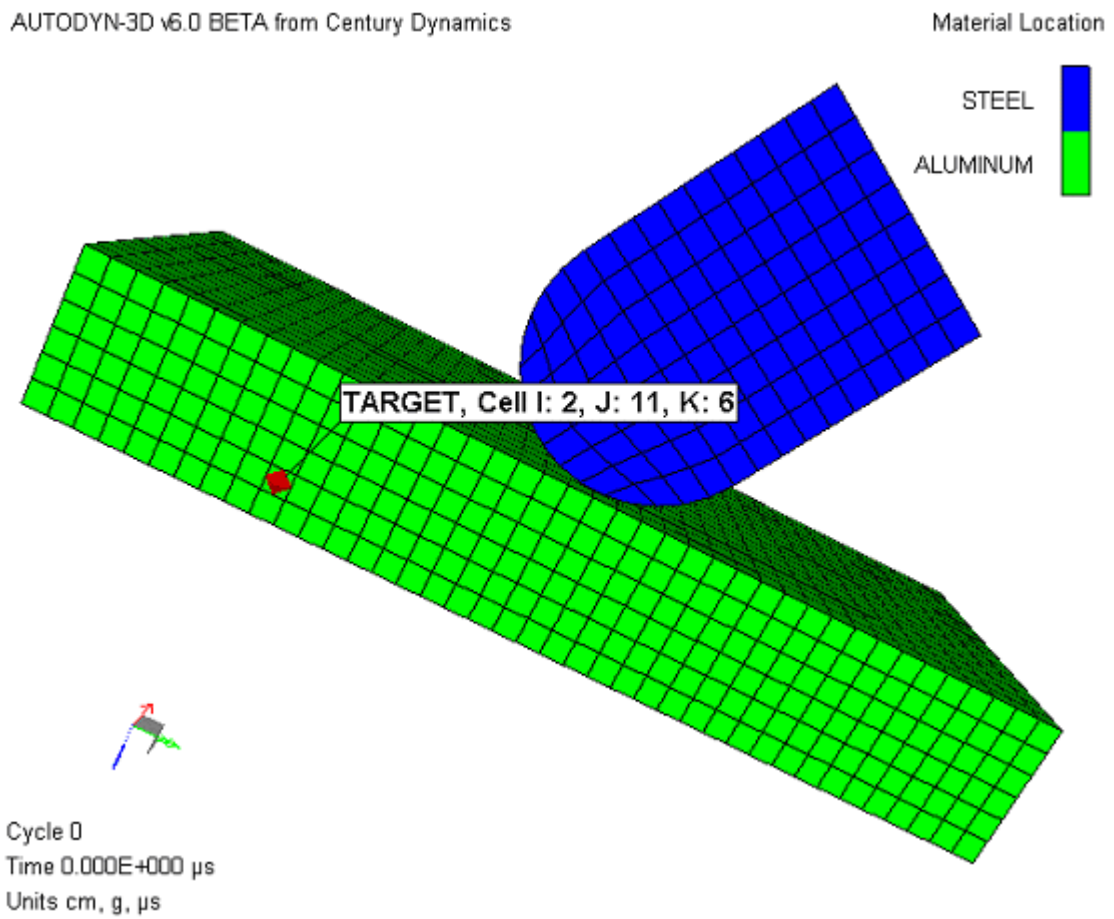
Settings - Marker





The Marker Plot lets you highlight a particular position, node or element of interest.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Specify Marker Position
Choose to specify the marker position either by XYZ coordinates or by specifying a part and I/J/K values for a node or element.

XYZ coordinates
Enter the coordinates of the marker location.

Node/Element/Face
Lets you highlight a particular node, element or face of an element.

Parts List
Select the part containing the node, element or face to be highlighted.

IJK
Select the I, J, and K values for the node, element or face of interest in case the part selected is structured.

Node Number/Element Number/Face Number
Enter the node, element for face number of interest in case the part selected is unstructured. Depending on the selection of User No. or Internal No. the node, element or face number displayed is the original User Number or the AUTODYN internal number.

Text labels

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box to label each marker

Text size
This defines the text size used for the label

Label bounds

Check this box to put a bounding box around each label.

Label offset
Check this box to set your own offset position for labels.

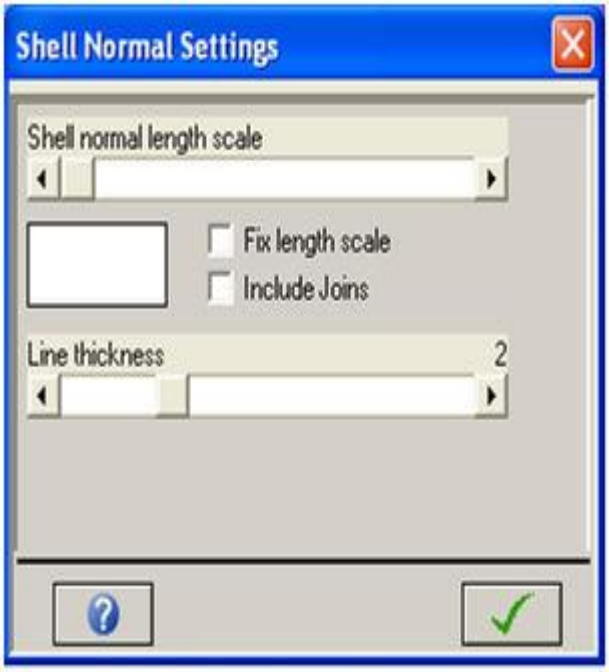
Leading line
Check this box if you want to show lines joining each marker to its label.

Radial offset
Check this box if you want to use radial label offsets instead of linear offsets.

X / Y / Z offset
These sliders let you adjust the label offset distance in the X, Y and Z directions.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Settings - Shell Normal



This window lets you change settings for plotting shell normals.

Local Axes length scale
Use this slide to vary the length of the arrows displaying the shell direction.

Fix length scale
Check this box to fix the shell normals length scale

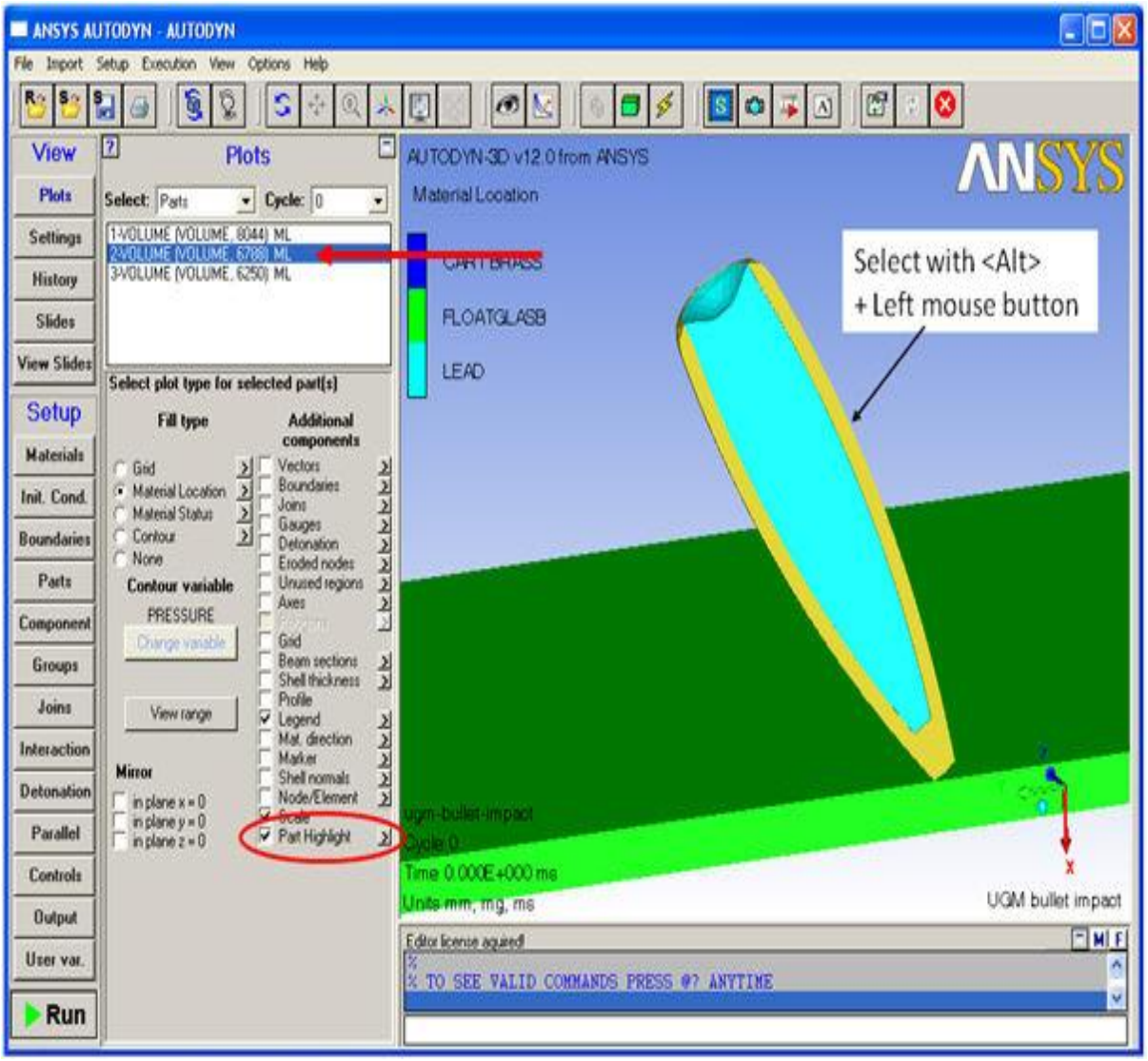
Include Joins
Select this toggle to show an average normal for joined nodes. If this toggle is unselected, a normal will be shown for each of the nodes joined.

Line thickness
Use this slider to vary the thickness of the lines displaying the shell normals.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

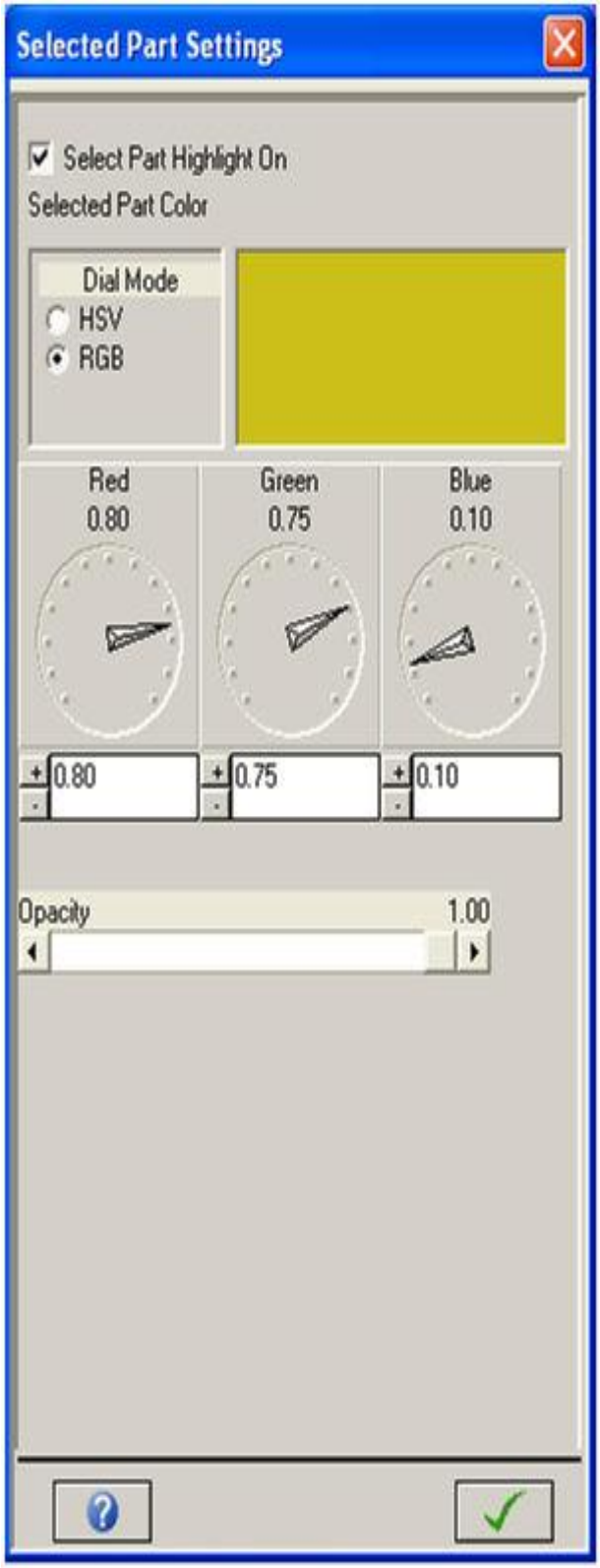
Settings-Part Highlights

When this option is selected, Parts selected in the plots panel will be highlighted on the displayed model. Selecting multiple parts in the parts list will highlight all the selected Parts.



Using <alt> + Left Mouse button you may select a part in the model view window and make it the current part. The part selected with the mouse will be selected in either the part list in the plots panel or parts panel. This is useful to identify parts and to make a particular part the current part active for zoning, filling or other part operations.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

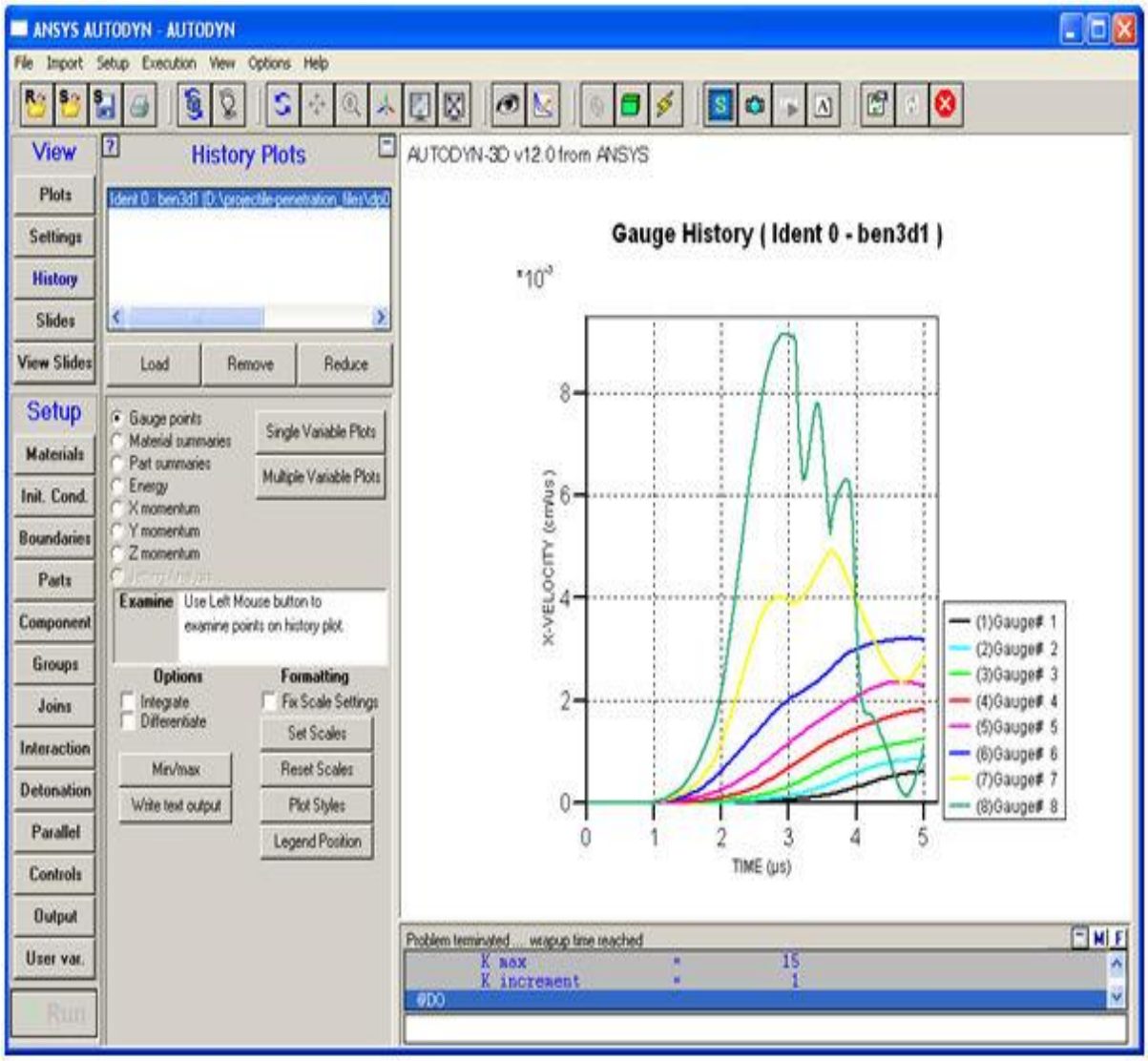


The default highlight color is golden yellow; you may alter the color and opacity of the highlight color that is applied to selected parts using the associated options panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History

History



This panel lets you compose and display time histories of Gauge points and Summary variables.

When you select this panel, the history and summary data associated with the model you are working with will be automatically loaded and the model's Ident will be displayed in the window at the top of the panel.

Load

Press this button to load history data for other models.

Remove

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press this button to remove history data for the selected model (at least one model must remain loaded at all times).

Reduce

Press this button to reduce the amount of history data stored for a model.

History Type

Select the type of history you want to view from the following options :

Gauge Points	Variable histories at gauge points
Material Summaries	Mass, energy and momentum sums for each material
Part Summaries	Mass, energy and momentum sums for each Part
Energy	Energy balance
X momentum	X momentum balance
Y momentum	Y momentum balance
Z momentum	Z momentum balance
Jetting Analysis	Jetting analysis plots (only active if model uses Jetting)

Single Variable Plots

Press this button to compose plots of a single variable

Multiple Variable Plots

Press this button to compose plots including multiple variables

Examine

Use the Left Mouse Button to select a point on the graph. The corresponding curve label and x and y values are displayed in the Examine panel.

Integrate

Check this box to view integrals of the variables in the current plot.

Differentiate

Check this box to view differentials of the variables in the current plot.

Min/Max

Press this button to view maximum and minimum values for the current plot.

Write textfile (uhs)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Output the data from the current plot (Write UHS for current history plot) or output all the gauge data from a history file (Write Output for all gauges and variables in selected ident).

Set scales

AUTODYN automatically chooses scales for your plots based on minimum and maximum values of the data. Pressing this button lets you set your own scales.

Plot Styles

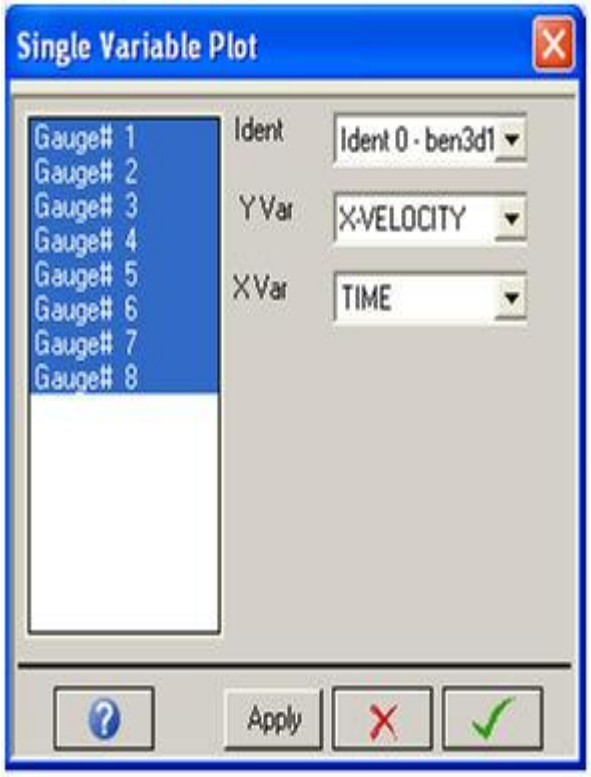
Press this button to set the color, style and thickness of lines for each plot

Legend Position

Press this button to position the legend anywhere in the view.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History - Single Variable Plot



This window lets you compose a single variable plot.

Single variable plots can contain one or more plots (histories) for the same X and Y variables.

Ident

This pull-down menu lets you select an Ident from the models you have loaded.

Y Var

This pull-down menu lets you choose the variable assigned to the Y axis.

X Var

This pull-down menu lets you choose the variable assigned to the X axis (most often TIME).

Histories to display

From the list on the left choose as many histories as you wish to display in the plot (use the **control** and **shift** keys in the usual way to make your selection).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The choices you have available for variables and histories will depend on the type of plot you have chosen (Gauge points, Material Summaries, etc.).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History - Reduce

It is usual to save history data very frequently (the default is every cycle) in order to capture high frequency responses.

If a calculation is run for a large number of cycles the history files can become very large.

This window lets you reduce the size of history files by removing data for some cycles.

Select file

In this box, select the ident of the file you wish to reduce in size.

Reduce by

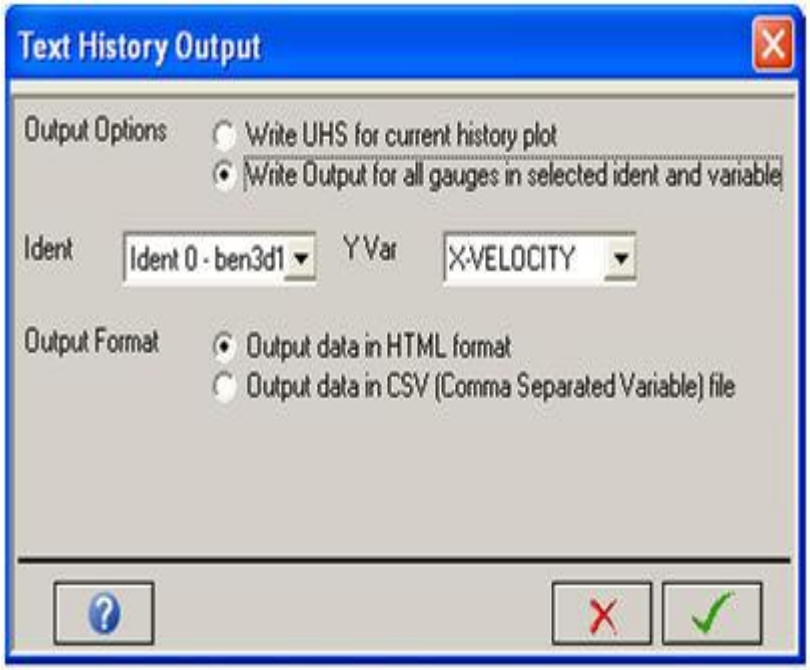
Specify the reduction factor in this field.

If you specify a reduction factor of 2, data for every other cycle will be removed from the file.

If you specify a reduction factor of 3, data for two out of every three cycles will be removed from the file.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History Write Text Output


Text History Output
Output Options
Write UHS for current history plot
Write Output for all gauges in selected ident and variable
Ident
Ident 0 - ben3d1
Y Var
X-VELOCITY
Output Format
Output data in HTML format
Output data in CSV (Comma Separated Variable) file

This dialog lets you output the data from the current plot (Write UHS for current history plot) or output all the gauge data from a history file (Write Output for all gauges and variables in selected ident).

Write UHS for current history plot

Select this option to output to a CSV (comma separated variable) file, which is suitable for importing into Microsoft Excel the data from the currently plotted history file.

Write Output for all gauges and variables in selected ident

Selecting an ident and variable to output, all the data for each gauge for the selected variable and ident is written to the output file.

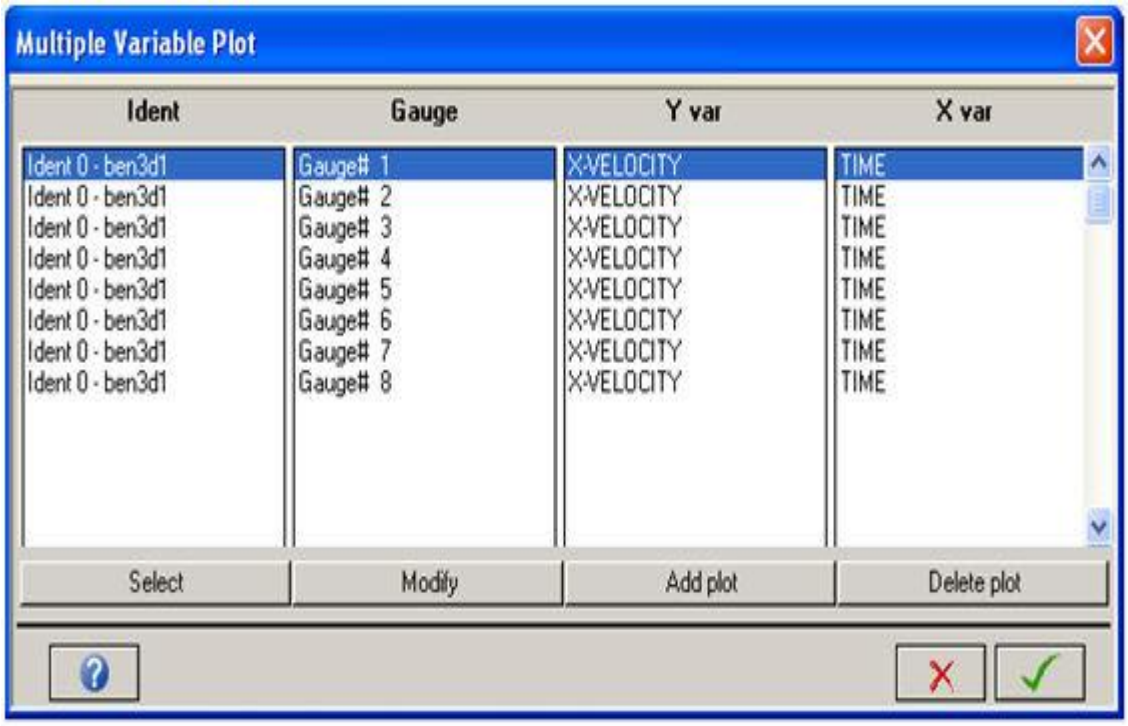
Output Format

Select the format for the data output, either a CSV (Comma separated variable) file or an html file.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Multiple Variable Plot

History - Multiple Variable Plot



This window lets you compose a multiple variable plot.

Multiple variable plots can contain one or more plots (histories), each with different variables for the X and Y axes.

This window list the plots (histories) you have currently define, displaying for each one its Ident, Gauge number, X variable and Y variable.

Below this list are four buttons :

Select

Press this button to quickly add multiple plots to the list.

Modify

Select a plot in the list and press this button to modify its settings.

Add plot

Select a plot in the list and press this button to add a new plot to the list. The initial settings for the new plot will be the same as the selected plot, but you can modify these values immediately in the dialog window that opens.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

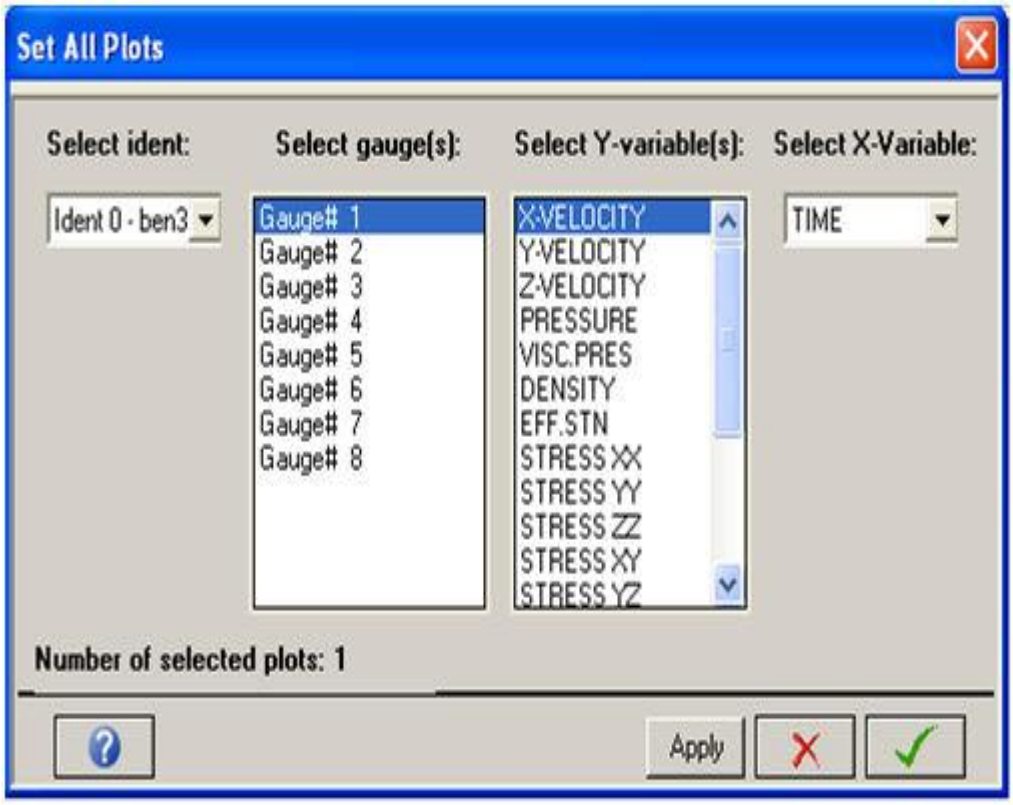
Delete plot

Press this button to delete a plot from the list.

The choices you have available for gauges and variables will depend on the type of plot you have chosen (Gauge points, Material Summaries, etc.).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History - Multiple Variable Plot - Set All Plots



This window lets you quickly add multiple plots to a multiple variable plot list.

Select ident

This pull-down menu lets you choose an ident from the models you have loaded.

Select gauge(s)

This box lets you choose one or more gauges (use the control and shift keys in the usual way to make your selection).

Select Y-variable(s)

This box lets you choose one or more variables for the Y axis (use the control and shift keys in the usual way to make your selection).

Select X-variable

This pull-down menu lets you choose a variable for the X axis.

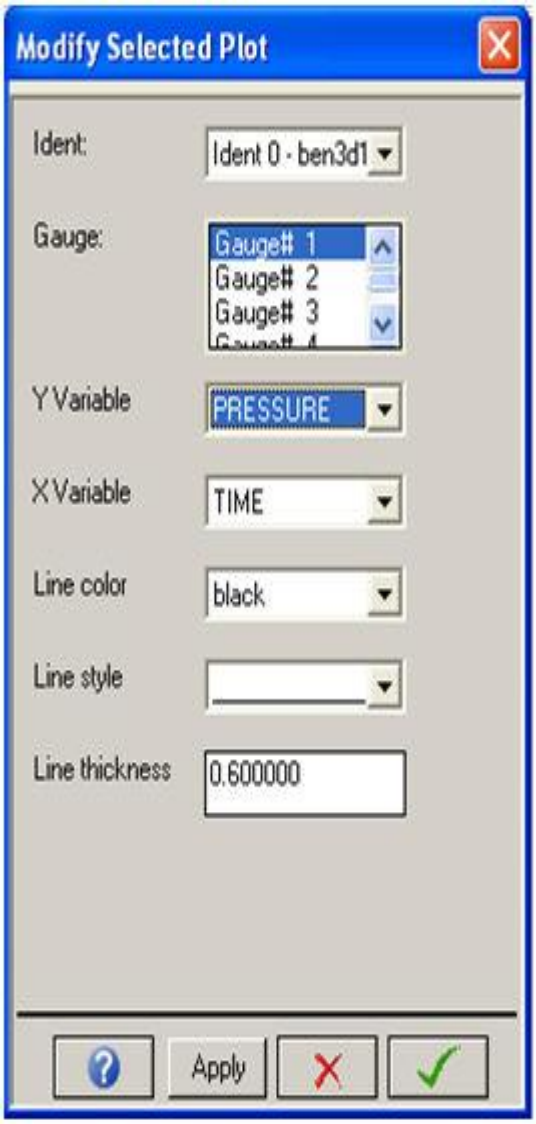
Number of selected plots:

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This tells you how many plots you have defined with your current selections. The number of plots is the union of all selections in the four input regions.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History - Multiple Variable Plot - Modify Selected Plot



This window lets you set or modify parameters for a plot to be included in a multiple variable plot.

Ident

This pull-down menu lets you choose the Ident for the plot from the models you have loaded.

Gauge

This box lets you select the gauge to be plotted

Y Variable

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This pull-down menu lets you pick the Y Variable

X Variable

This pull-down menu lets you pick the X Variable

Line color

This pull-down menu lets you pick the color of the line.

Line style

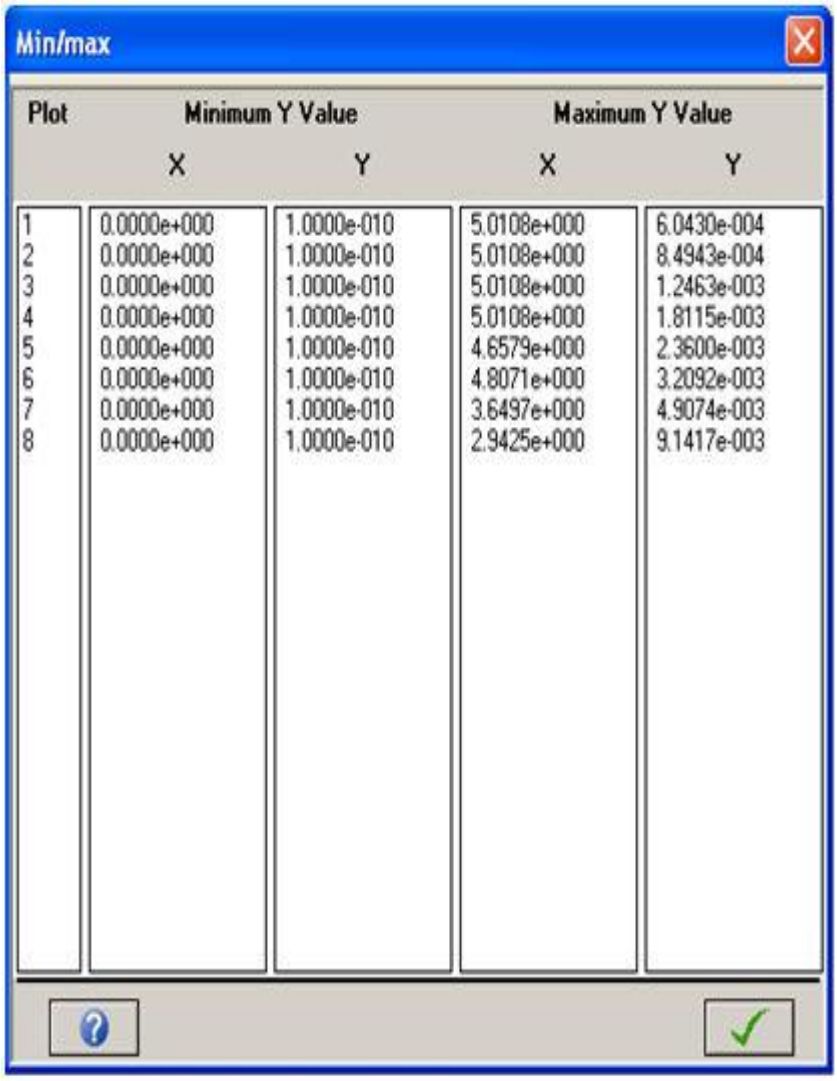
This pull-down menu lets you pick the line style (solid, dashed, etc.)

Line thickness

This box lets you set the line thickness

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History - Min/Max



This window displays the maximum and minimum values for each history plot shown in the current view.

X min is the X value at which the minimum Y value occurs

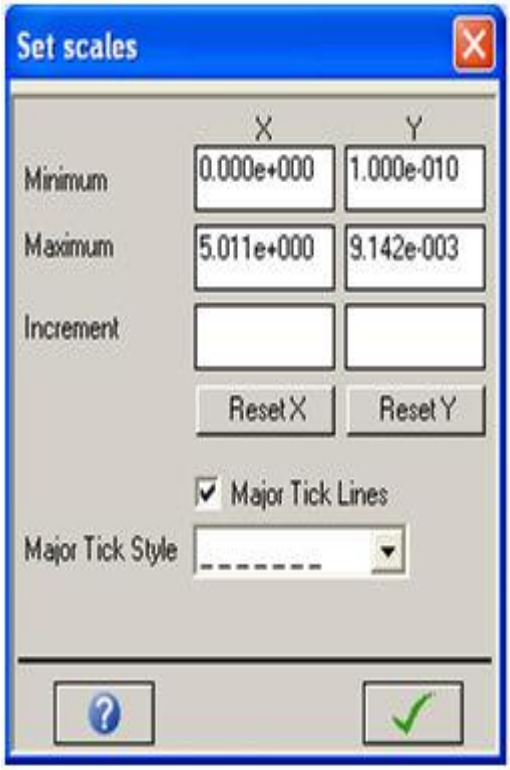
Y min is the minimum Y value

X max is the X value at which the maximum Y value occurs

Y max is the maximum Y value

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History - Set Scales



This window lets you set your own scales for history plots.

Minimum

These fields let you set the minimum values for your X and Y scales

Maximum

These fields let you set the maximum values for your X and Y scales

Increment

These fields let you set the X and Y increments where tick marks will be placed on the scales.

Reset X

Press this button to reset the maximum and minimum X values to those calculated automatically by AUTODYN.

Reset Y

Press this button to reset the maximum and minimum Y values to those calculated automatically by AUTODYN.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Major Tick Lines

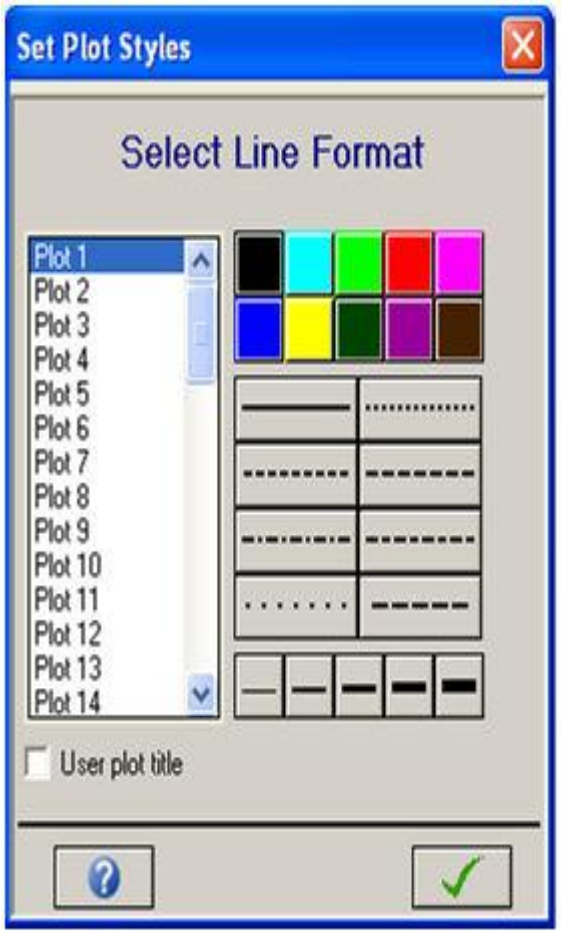
Check this box if you want major tick lines to extend from left to right and top to bottom across your plot.

Major Tick Style

This pull-down menu lets you choose the line style used to drawn tick lines (solid, dashed, etc.)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History - Set Plot Styles



This window lets you set the color, style and thickness of lines for each plot

Plot Number

Plots are numbered in the order you specify them, starting at 1. The box on the left lets you select a plot number whose style you wish to set.

Color

Select a color for the selected plot from the top two rows on the right.

Line Style

Select a line style for the selected plot from the next four rows on the right

Line Thickness

Select a line thickness for the selected plot from the bottom row on the right

User plot title

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box if you want to supply your own title for the plot. The default title is "History Plot"

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

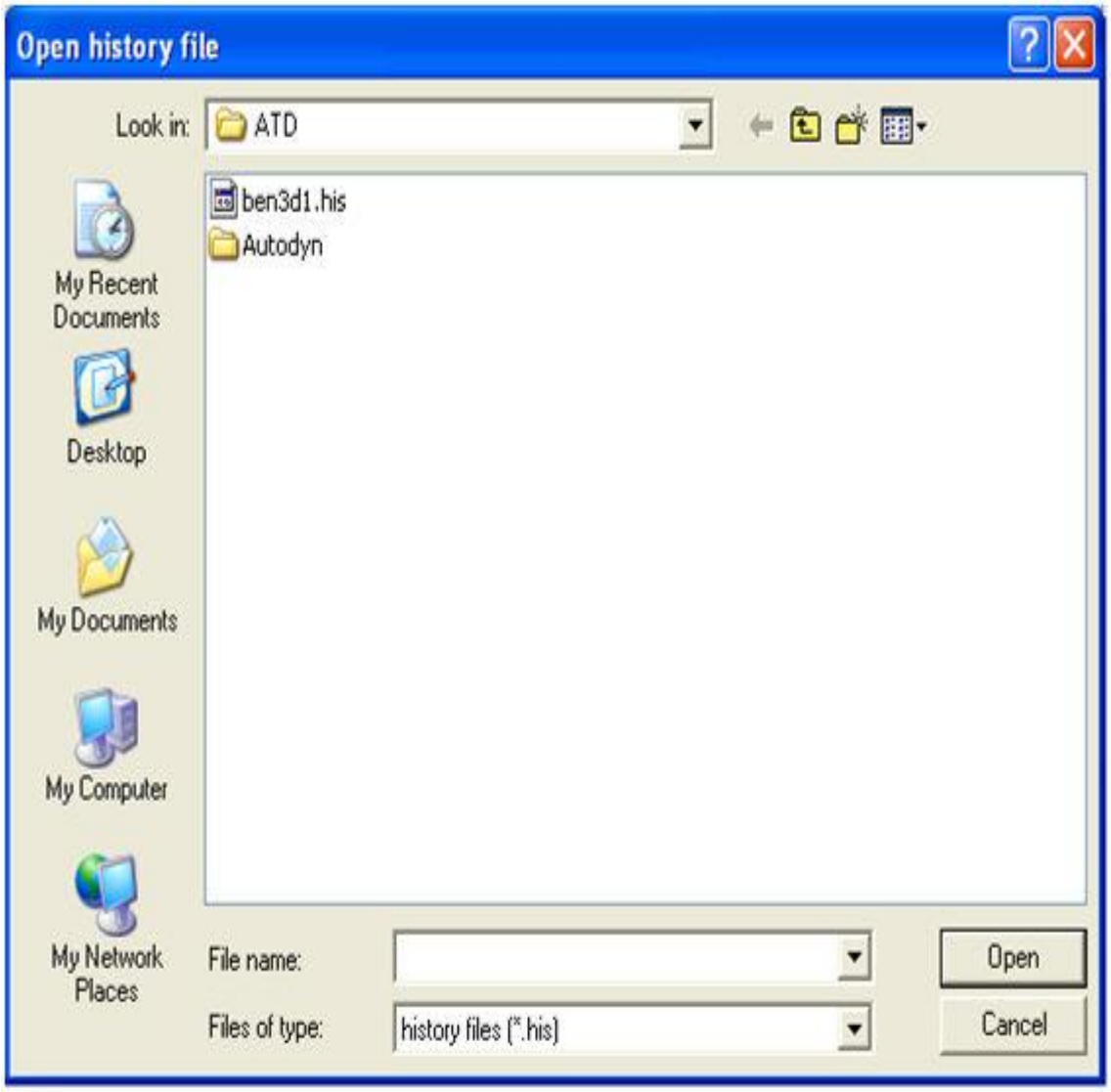
History - Set Legend Position

This window lets you place the plot legend anywhere in the view.

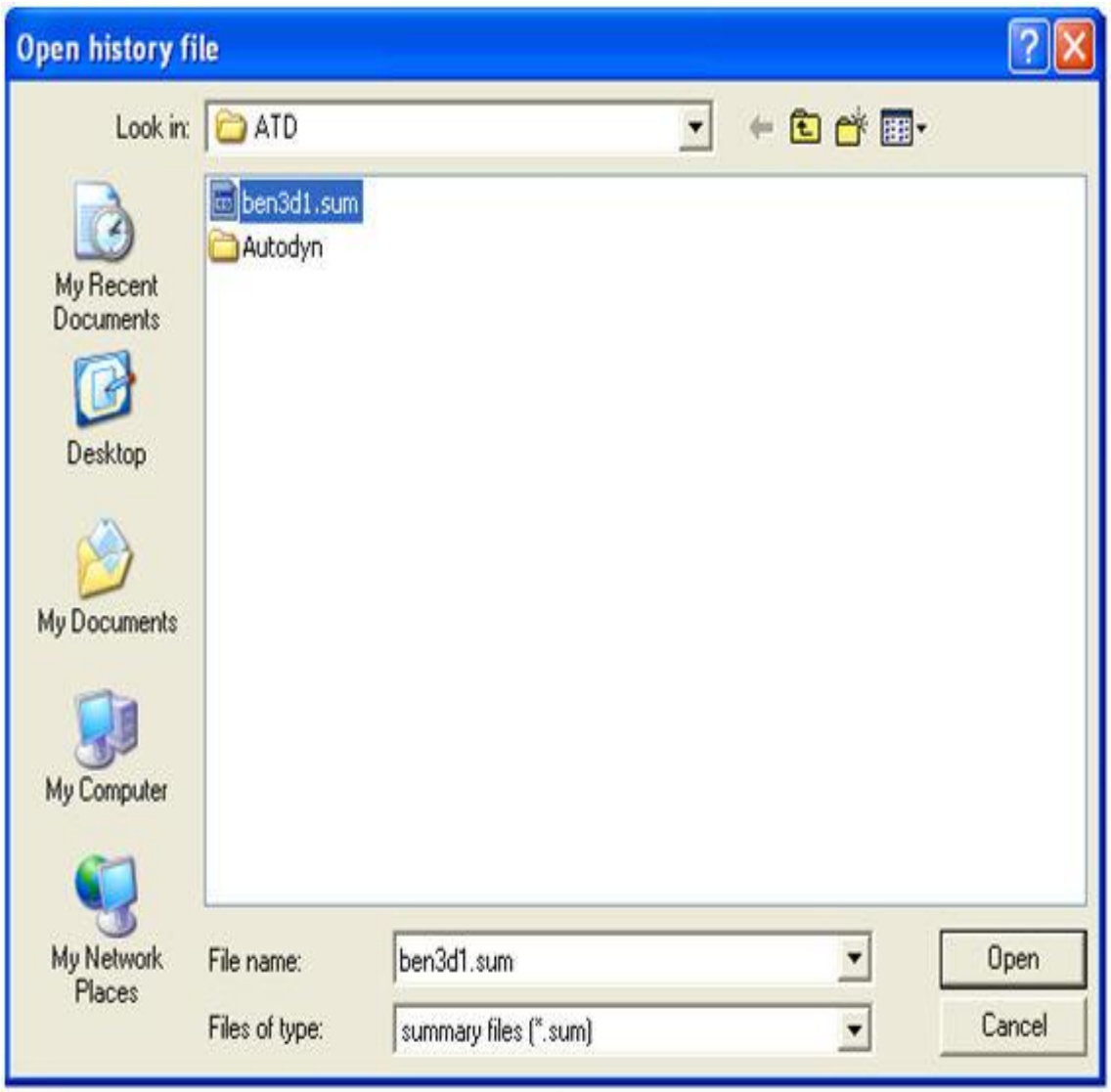
Use the two sliders to adjust the horizontal and vertical position of the legend until it's where you want it.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

History - Load



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



This window lets you load history files for plotting.

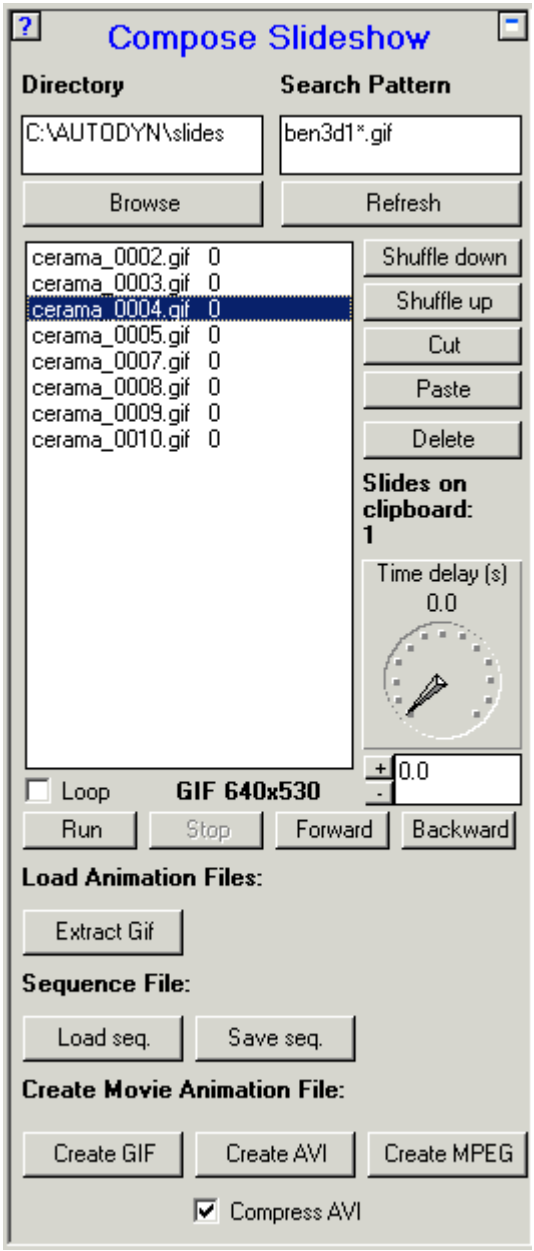
History data is stored in two types of files.

.his These files contain history data for the gauge points defined for a model.
.sum These files contain history data for summary plots (Material Summaries, Part Summaries, Energy, etc.)

Depending on the plot type you have chosen, this window lets you navigate your file system to find **.his** or **.sum** files for loading.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Slides



This panel lets you compose slide shows from images created using the **Text Slide**, **Camera** (single slide) and **Animation** (multiple slides) buttons on the Tool Bar.

Directory

The current directory where the slides are stored and processed is shown in this Box.

Press the **Browse** button below the box to change this directory.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

After changing the directory, press the **Refresh** button to update the list of slides in the slide sequence box.

Search Pattern

You can type a search pattern in this field that restricts the files you see in the slide sequence box.

In the example above the search pattern restricts files you see to those having an ident "ben3d1" and extension **.gif** .
(all **.gif**, **.bmp, .tiff** and **.jpeg** files are shown if the search pattern does not include an extension)

Slide Sequence Box

The left center box shows the current slide sequence.

Initially the slide sequence will contain all the files in the current **Directory** that satisfy the **Search Pattern,** listed in alphanumeric order.

You can use the **Cut/Paste** and **Shuffle Up/Down** buttons to remove or rearrange files in the slide sequence.

Press the **Refresh** button if you want to abandon your current sequence and start over with all files in alphanumeric order.

When you select a file in the slide sequence, the image of that file is automatically displayed in the View panel.

Shuffle Up/Down

These buttons let you move one or more files up or down in the slide sequence.

Select the group of files you want to move (using the shift key to select more than one file), and press the **Shuffle Up** or **Shuffle Down** button to move them collectively one position up or down respectively.

Cut

Select one or more files in the slide sequence and press this button to remove them. The removed files are placed on the clipboard and you can use the **Paste** button to restore them at some other point in the sequence. If you use **Cut** a second time, the clipboard is cleared and the newly removed files are placed on it
.

Paste

Press this button to insert all files from the clipboard **after** the selected file in the slide sequence.

Slides on clipboard

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This shows you how many slide files are currently on the clipboard.

Time delay

Select one or more files in the slide sequence and rotate the dial to set the time delay you want to apply to them (a slide's time delay is the time, in seconds, for which it is shown).

Instead of using the dial, you can alternatively type the time delay in the field below the dial.

Loop

Check the box if you want your slide show to replay automatically.

Run

Press this button to play the current slide sequence. The entire sequence is played, irrespective of which files are currently selected.

Stop

Press this button to stop playing the current slide sequence.

Forward / Backward

These buttons let you quickly change your selection to the preceding (**Backward**) or following (**Forward**) file in the slide sequence.

Since this procedure automatically updates the View panel, this is a good way of viewing your slides manually.

Extract GIF

Press this button to extract individual .gif files from an animated GIF and place them in the current directory.

Warning!
This option removes the current slide sequence and replaces it with a sequence containing only the newly extracted files (irrespective of the current search pattern).

Save Seq.

Press this button to store the current slide sequence in a sequence file.

Load Seq.

Press this button to load a sequence stored in a sequence file and apply it to the current directory.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Warning!
If the files listed in the sequence do not exactly match those in the current directory, results may be unpredictable.

Create GIF

Press this button to create a GIF animation from the current slide sequence.

Create AVI

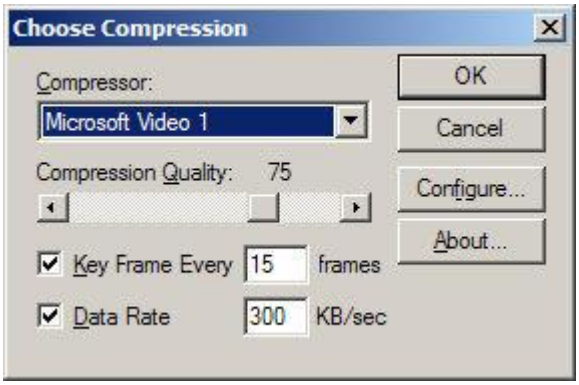
Press this button to create an AVI animation from the current slide sequence.

Create MPEG

Press this button to create an MPEG animation from the current slide sequence.

Compress AVI

Check this toggle to choose compression for AVI.



In the choose compression dialog, select the type of compression to use for the movie. The size and quality of the resulting AVI animation will depend on the choices made in this panel.

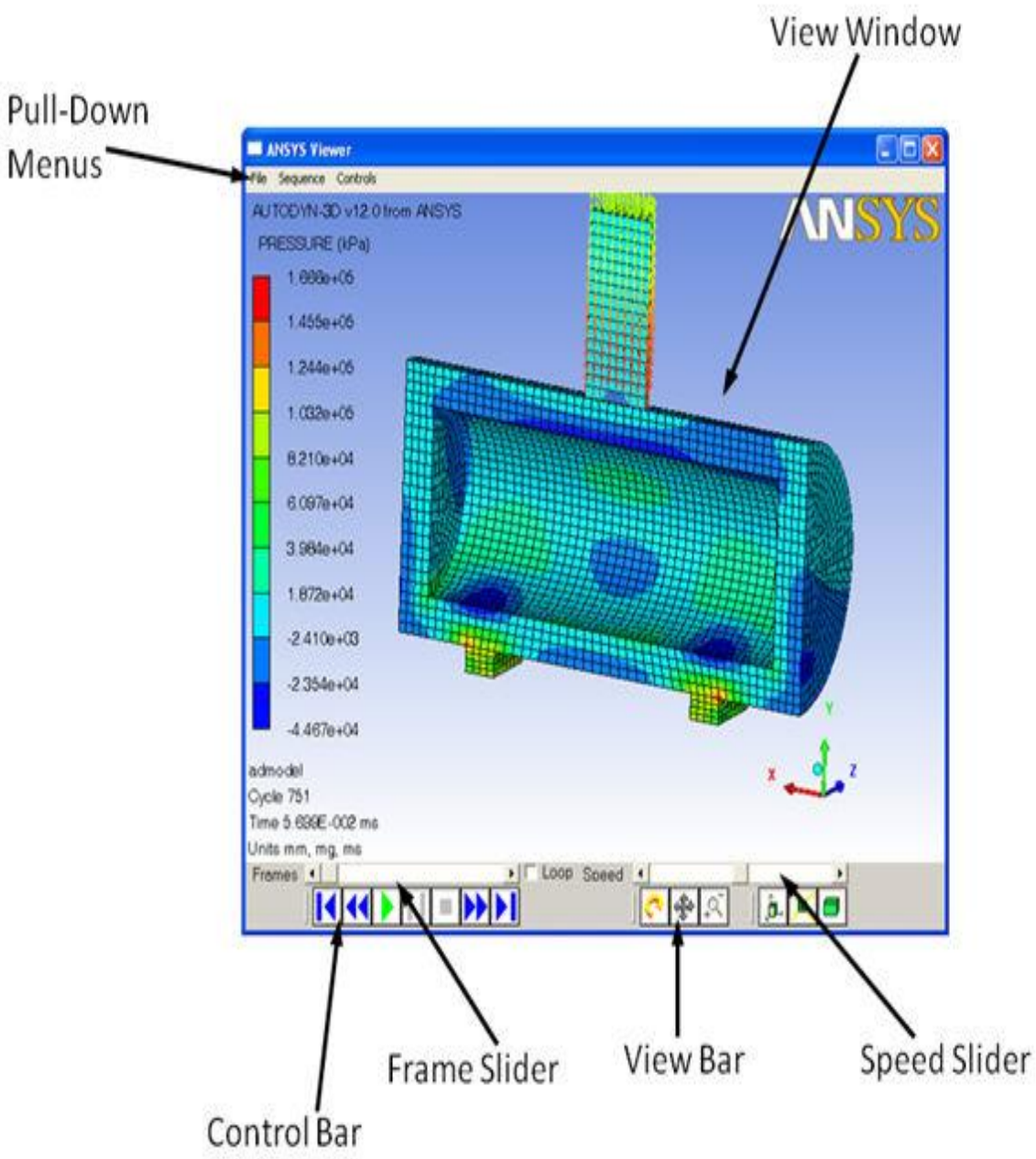
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

View Slides (Slide Viewer)

The View Slides option on the Navigation Bar starts the Slide Viewer (it can also be run independently of AUTODYN).

The Viewer lets you load and view GIF and GFA animations.

Components of the Viewer's user interface :



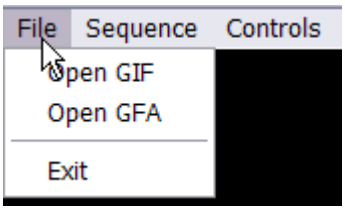
View Window

This is where your animations are shown.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

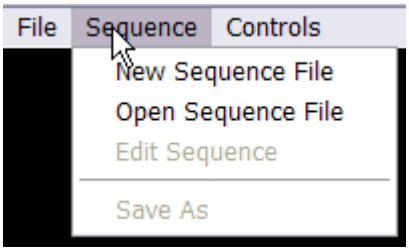
Pull-down Menus

File



This pull-down menu lets you load single GIF and GFA animation files for playback, and exit the program.

Sequence



This pull-down menu lets you playback multiple GIF animations using sequence files.

A sequence file contains a list of GIF animations you want to view consecutively.

All the animations in a sequence file must be in the same directory as the sequence file.

New Sequence File

This option opens a new sequence file.

Open Sequence File

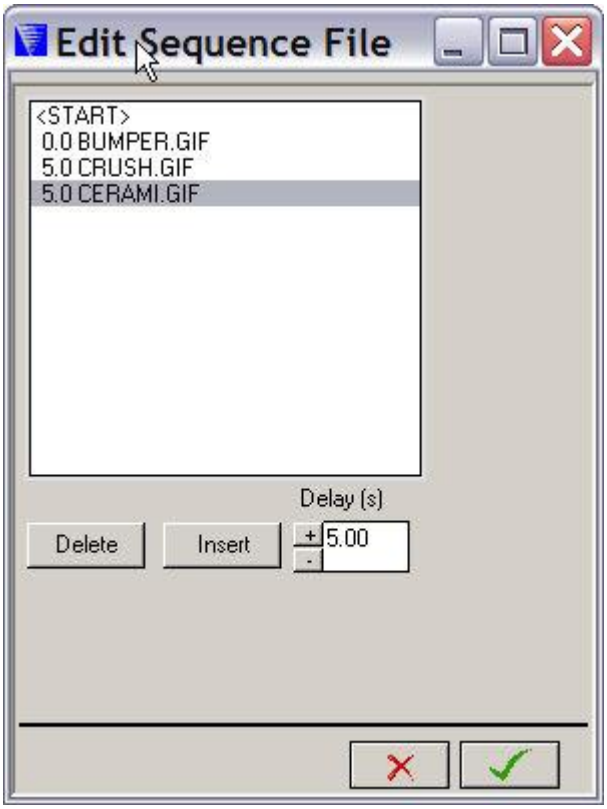
This option opens an existing sequence file.

Edit Sequence

This option lets you insert and delete animations in the current sequence file.

The following dialog window appears when you select this option :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Sequence list

The box at the top of this dialog contains a list of the animations currently included in the sequence, together with their time delay (how long the first slide in the sequence is shown, in seconds).

Insert

Press this button to insert another animation immediately after the selected animation.

Delete

Press this button to delete the selected animation.

Delay

Enter a time delay in this field (in seconds). When you **Insert** an animation, this time delay is applied.

You can create and edit sequence files manually if you wish. Sequence files must have a **.seq** extension and use the following format

Format	**Example**
The number of animations in the sequence	3

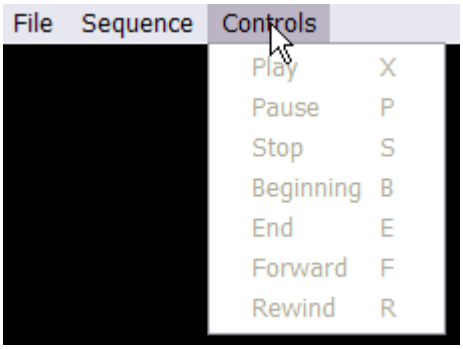
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

	0.0 anim.gif
Default frame delay in seconds + <space> + file name	1.2 another.gif
Default frame delay in seconds + <space> + file name	1.6 yetanother.gif
Default frame delay in seconds + <space> + file name	
etc.	

Save As

This option saves the current sequence file (**.seq**)

Control



This pull-down menu provides the same functions that are available in the Control Bar.

Control Bar

This bar lets you control the playing of the GIF or GFA animation you have loaded.

Pressing a button on this bar performs the following actions (you can type the corresponding **Hotkey** as an alternative to pressing the a button).

Button	Hotkey	Action
	b	Go to the first frame of the animation
	r	Go back one frame
	x	Play (from the current frame)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

p	Pause (at the current frame). This is a toggle button.
s	Stop (at the current frame)
a	Advance one frame
e	Go to the last frame of the animation

View Bar

This bar only appears when you have loaded a GFA animation.

The buttons on this bar let you control how you view your GFA animation and work in the same way as the corresponding icons in the AUTODYN **Tool Bar**

Rotate view
Translate view
Zoom in or out
Set view (brings up a window that allows you to set an exact view)
Reset view (to the initial view)
Turn perspective on or off (toggle)

Frames Slider

This slider lets you move quickly to any frame in the animation(s).

Speed Slider

This slider lets you adjust the playback speed for animations.

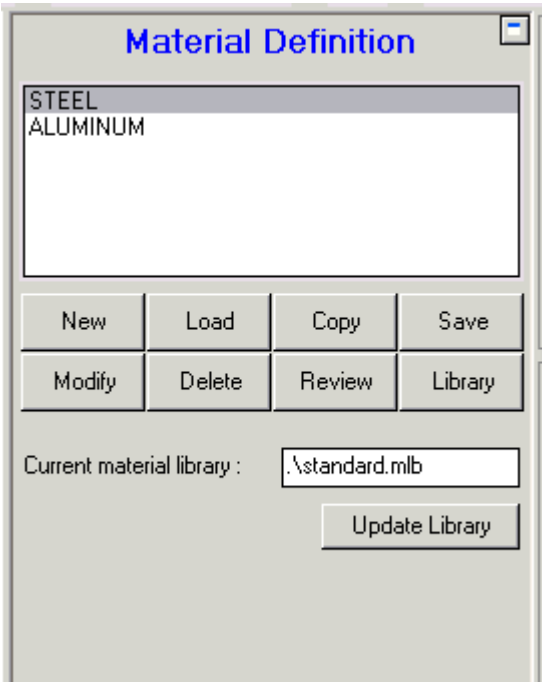
Loop

Check this box if you want the animation(s) to automatically start again from the first frame when the last frame is reached.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Materials

Materials



This panel lets you define the materials you want to use in your model.

Material List

The box at the top of this panel lists the materials you have currently defined for your model. You can select materials in this list.

New

Press this button to define a new material.

Modify

Press this button to modify parameters for the selected material.

Copy

Press this button to copy material component data from an existing material to either a new material or another existing material.

Delete

Press this button to delete one or more materials from your model.

Review

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press this button to review the parameters for the selected material in a browser window.

Library

The default material library is **standard.mlb**. This contains all the material data provided by Century Dynamics.

Press this button if you want to change the material library to some other material library you have obtained or created.

Load

Press this button to load materials from your current material library.

Save

Press this button to save the materials you have defined to a material library.

Current material library

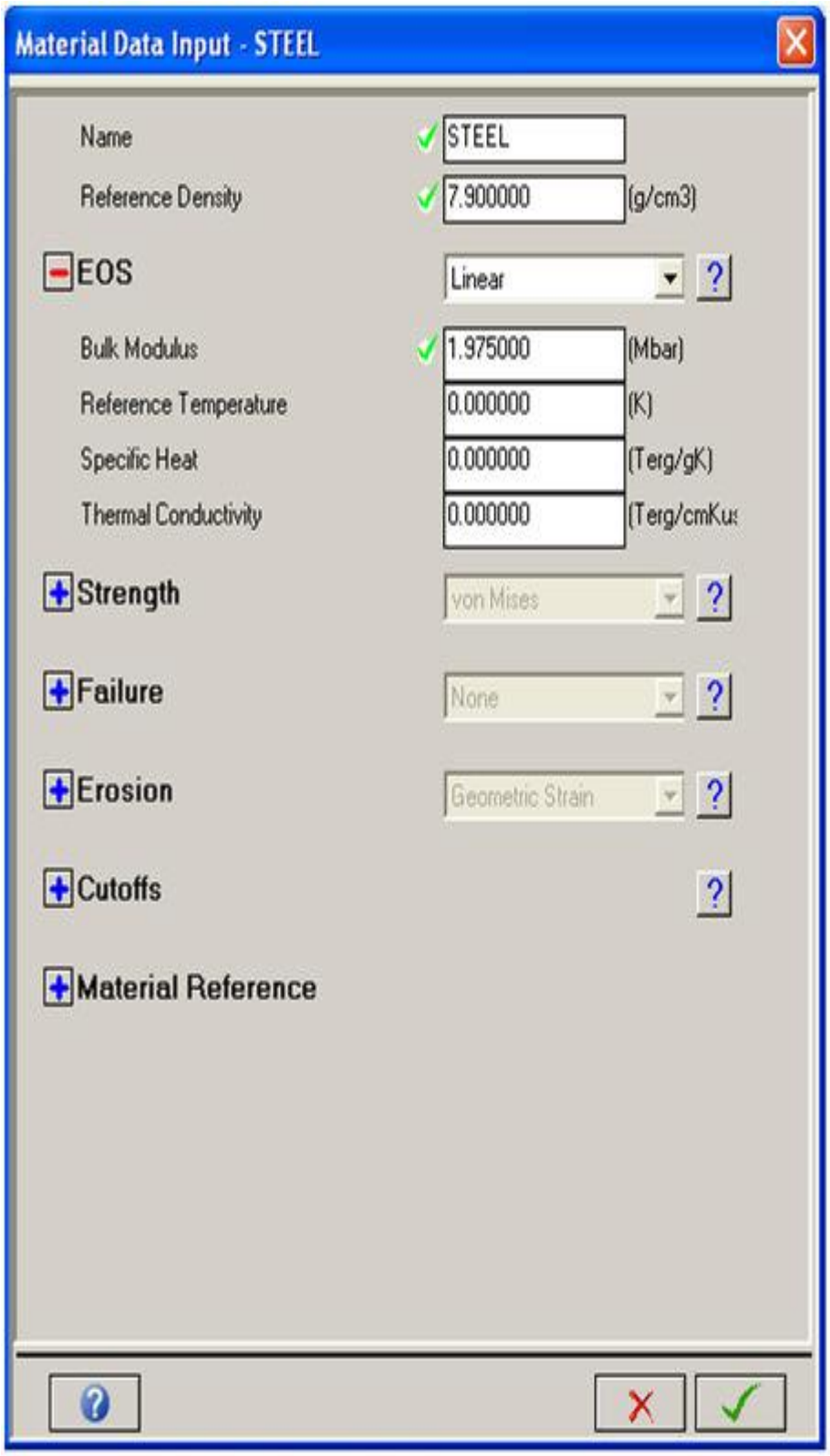
This box displays the current material library you have selected.

Update Library

Press this button to update old material library files, so that they can be used with this version of AUTODYN.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Materials - New / Modify



This window lets you define or modify the model components and parameters used for a material.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Name

Enter a name for your material in this field.

Reference Density

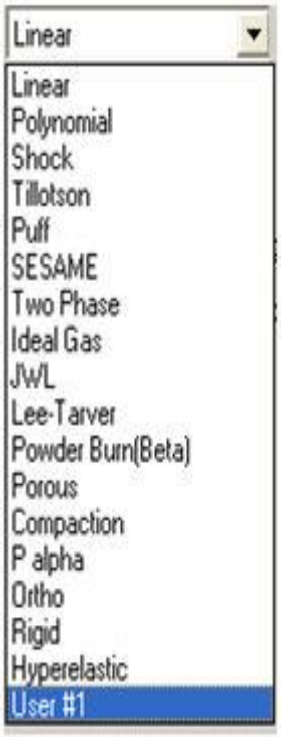
Enter the reference density (usually the initial density) for your material in this field.

EOS

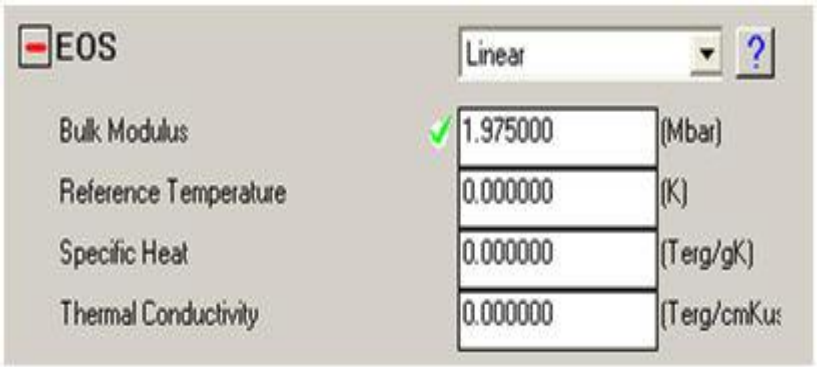
This option lets you define the Equation of State you want to use for your material.

Press + next to this option to expand the dialog.

Select the Equation of State you want to use from the pull-down menu :



Then type in parameter values in the fields provided :



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Strength

This option lets you define the strength model you want to use for your material.

Press + next to this option to expand the dialog.

Select the strength model you want to use from the pull-down menu :

Then type in parameter values in the fields provided:

Failure

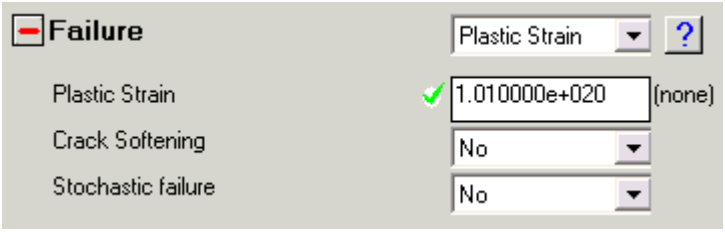
This option lets you define the failure model you want to use for your material.

Press + next to this option to expand the dialog.

Select the failure model you want to use from the pull-down menu :

Then type in parameter values in the fields provided :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



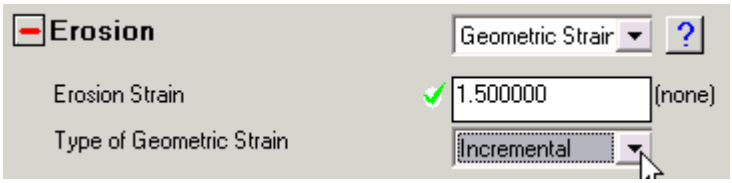
Erosion

This option lets you define the erosion model you want to use for your material.

Press + next to this option to expand the dialog.

Select the erosion model you want to use from the pull-down menu :

Then type in the parameter values in the fields provided :



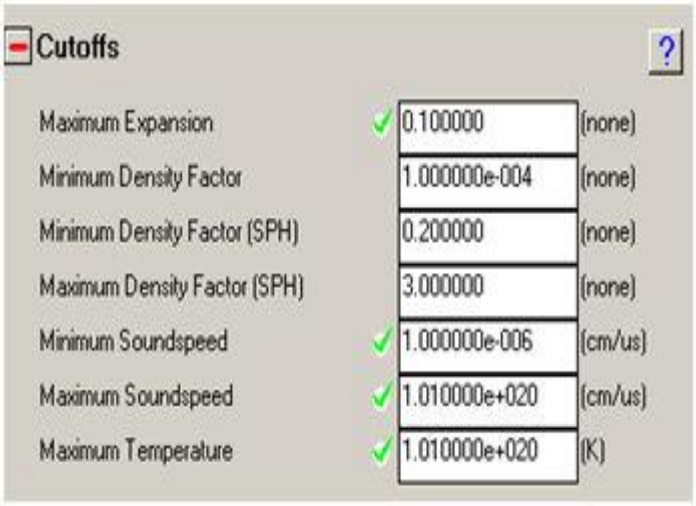
Cutoffs

This option lets you change the cutoffs for your material.

It is best to use the default values for these cutoff if you are using the default units (mm,mg,ms).

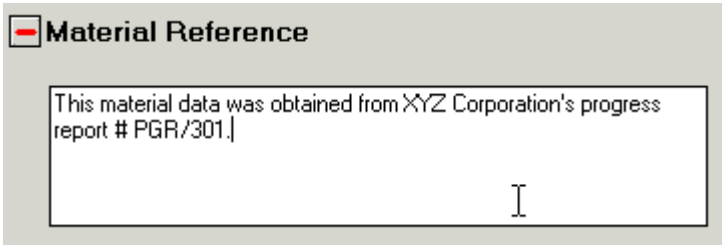
Press + next to this option to expand the dialog and change whatever cutoffs you want.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Material Reference

Press + next to this option to expand the dialog and type in a reference for your material in the field provided.



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Materials - Copy

This window lets you copy component data from an existing material to either a new material or another existing material.

Select material to copy from

Select the material you want to copy from this pull-down menu.

Select data components to copy

Check the box alongside the components you want to include in the copy process.

Copy to

Choose to copy the data to a new material or an existing material.

If you copy to a new material, components not copied will be set to the default state.

If you copy to an existing material, components not copied will retain their existing state.

New material name

If you choose to copy to a new material, enter your name for the new material in this field.

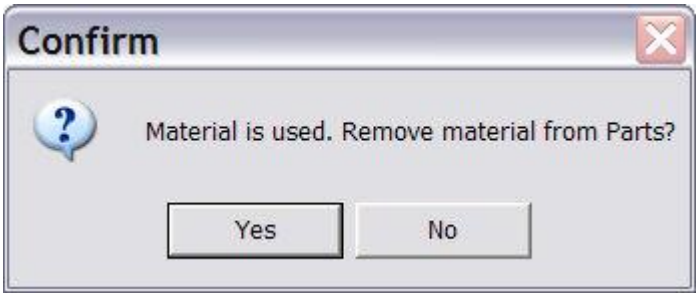
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Materials - Delete

This window lets you delete materials from your model.

Select the material(s) you want to delete and press .

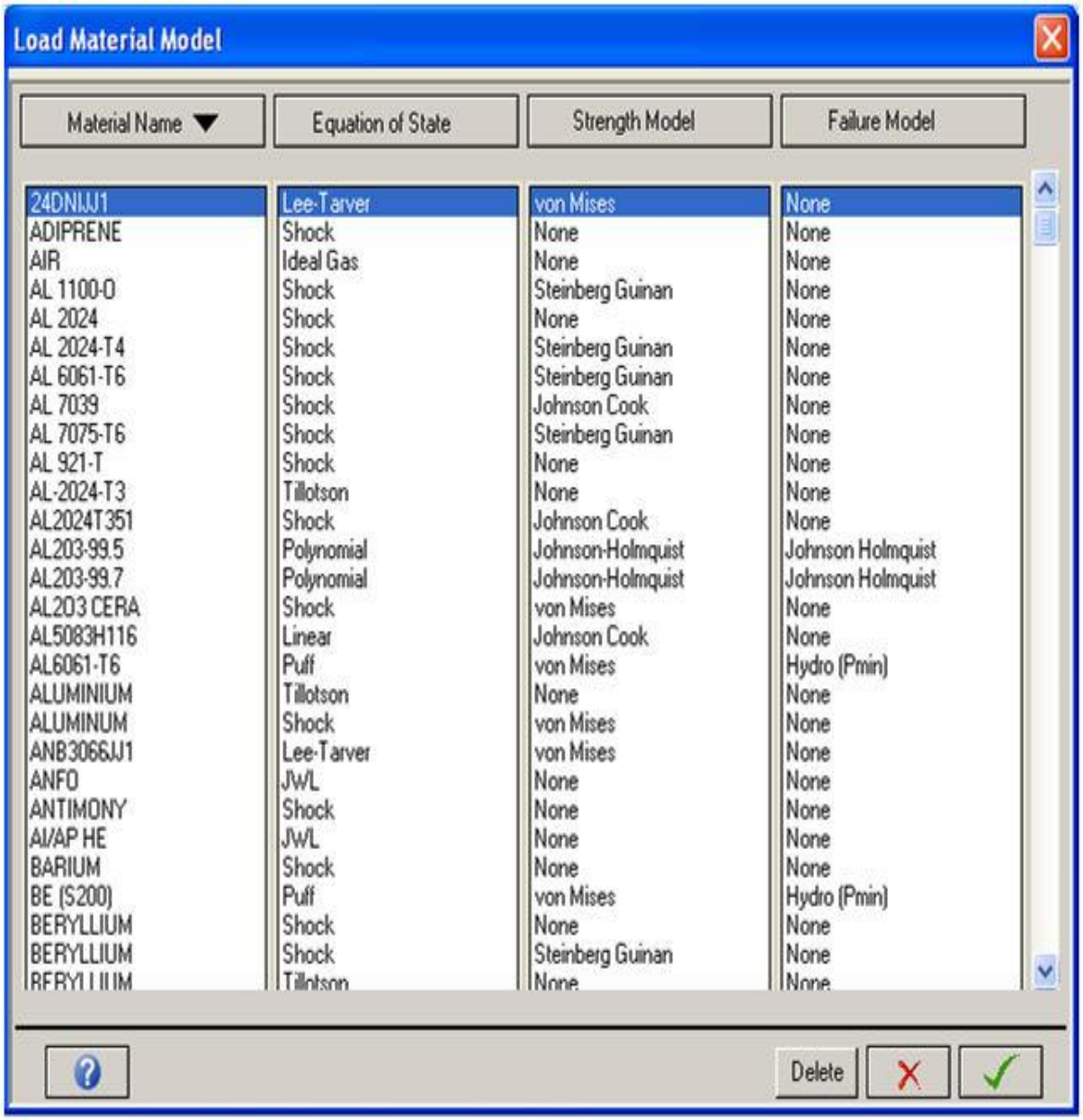
If the material you want to delete has been assigned to one or more Parts, you will be asked to confirm that it's OK to remove the material from the Part(s) :



Elements that have their material removed become **Unused**.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Materials - Load


Load Material Model
Material Name
Equation of State
Strength Model
Failure Model
24DNIJJ1 Lee-Tarver von Mises None
ADIPRENE Shock None None
AIR Ideal Gas None None
AL 1100-0 Shock Steinberg Guinan None
AL 2024 Shock None None
AL 2024-T4 Shock Steinberg Guinan None
AL 6061-T6 Shock Steinberg Guinan None
AL 7039 Shock Johnson Cook None
AL 7075-T6 Shock Steinberg Guinan None
AL 921-T Shock None None
AL-2024-T3 Tillotson None None
AL2024T351 Shock Johnson Cook None
AL203-99.5 Polynomial Johnson-Holmquist Johnson Holmquist
AL203-99.7 Polynomial Johnson-Holmquist Johnson Holmquist
AL203 CERA Shock von Mises None
AL5083H116 Linear Johnson Cook None
AL6061-T6 Puff von Mises Hydro (Pmin)
ALUMINIUM Tillotson None None
ALUMINUM Shock von Mises None
ANB3066JJ1 Lee-Tarver von Mises None
ANFO JWL None None
ANTIMONY Shock None None
AI/AP HE JWL None None
BARIUM Shock None None
BE (S200) Puff von Mises Hydro (Pmin)
BERYLLIUM Shock None None
BERYLLIUM Shock Steinberg Guinan None
Delete

This window lets you load materials from the current Material Library.

The available materials are listed in a vertical scroll box. For each material four columns provide its material name, Equation of State, Strength Model and Failure Model.

Press the button at the top of each column to sort materials alphanumerically in that column.

Using the control and shift keys as necessary, you can select one or more materials.

Press to load the selected materials into your model.

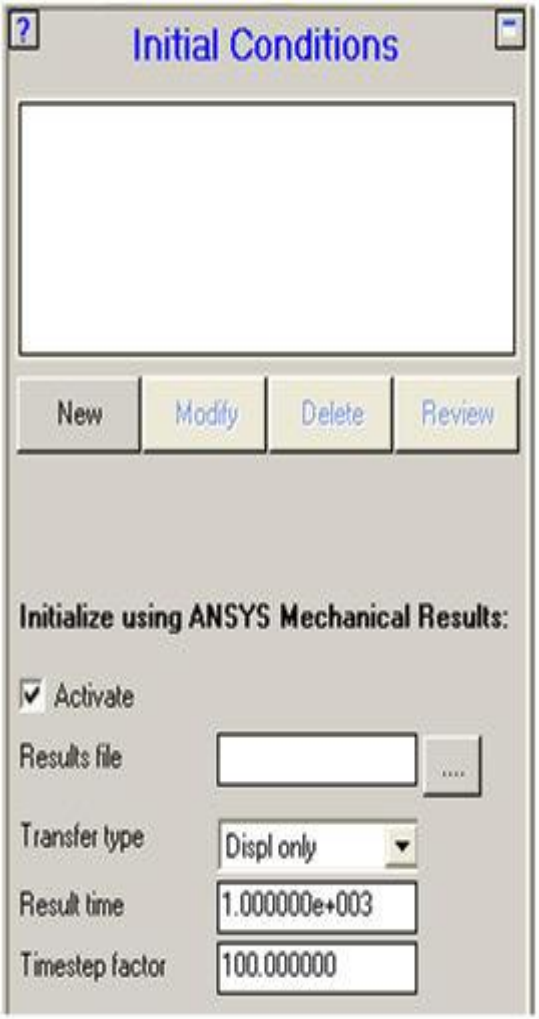
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press to delete the selected materials from the library.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Initial Conditions

Initial Conditions



This panel lets you create **Initial Condition Sets** that can be used in **Fill** operations for **Parts**.

You do not have to use an Initial Condition Set to fill a Part, but there is a major advantage in using one.

Any subsequent changes you make to an Initial Condition Set are automatically applied to the Parts that have been filled using that Initial Condition Set.

This means you don't have to refill the Parts (this is currently not true for Euler Parts).

Initial Condition Set List

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The box at the top of this panel lists the Initial Condition Sets you have currently defined for your model. You can select an Initial Condition Set in this list.

New

Press this button to define a new Initial Condition Set.

Modify

Press this button to modify the selected Initial Condition Set.

Delete

Press this button to delete one or more Initial Condition Sets from your model.

Review

Press this button to review the parameters for the selected Initial Condition Set in a browser window.

Implicit-Explicit Model Initialization

Activate

Select this option to initialize an AUTODYN analysis model using the results of an ANSYS Mechanical implicit analysis.

Results File

Select the ANSYS .rst results file to be used to initialize the AUTODYN analyses.

Transfer Type

Select the initialization method; either the Displacement Only or the Material State option as described below.

Results Time

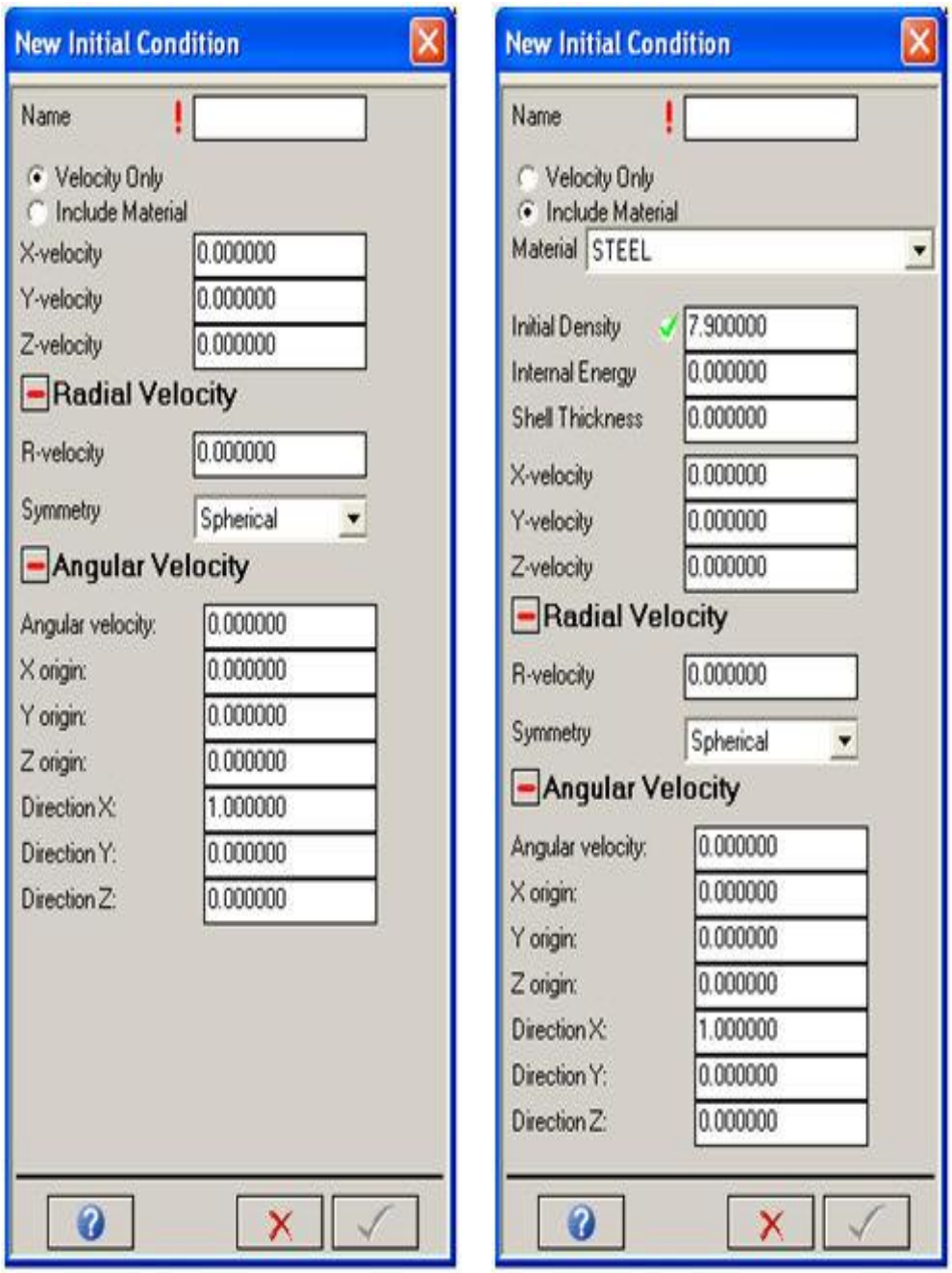
The result time from the ANSYS Mechanical implicit analysis at which to extract results from the .rst file.

Timestep factor

(Note that this option is only applicable for the Displacement Only method.) The initial timestep from the AUTODYN solution is multiplied by the timestep factor. The resulting time (PVELTM) is used together with the nodal displacements from the ANSYS Mechanical analysis to calculate constant nodal velocities. These nodal velocities are applied to the AUTODYN model over PVELTM in order to initialize the AUTODYN nodes to the correct positions.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Initial Conditions - New/Modify



This window lets you define a new Initial Condition Set.

Name

Enter your name for the Initial Condition Set in this field

Velocity Only

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select this option if you want your Initial Condition Set to only include velocities (top left window).

Include Material

Select this option if you want your Initial Condition Set to include velocities and a material and its associated variables (top right window)

Material

Use this pull-down menu to select the material you want to assign to the Initial Condition Set.

Material variables
Enter the appropriate values for the material density and internal energy to be used with this initial condition.

In addition, if the initial condition is to be used with a shell element, the element thickness can be specified. Note that this must be used for the unstructured solvers.

Translational Velocity
Specify the global x, y and z velocity associated with the initial condition.

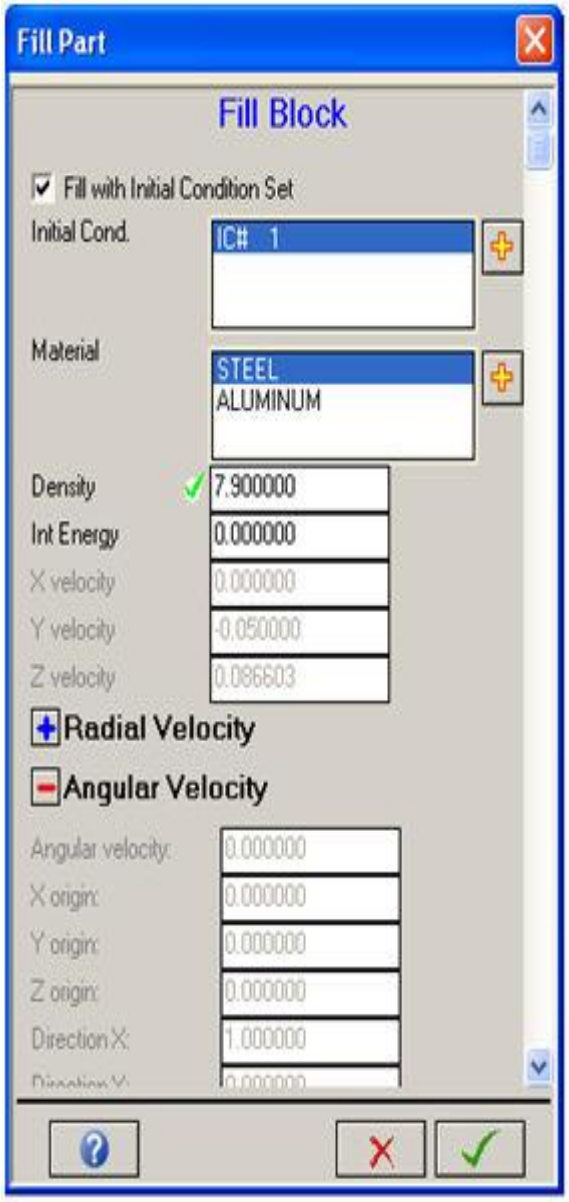
Radial Velocity
This option lets you define an initial velocity of fixed magnitude. The direction of the velocity will be defined by the radial position of each node from the original (0.,0.,0.).

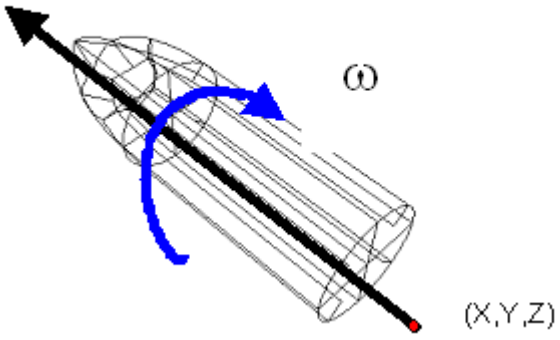
Spherical and cylindrical symmetry options are available.

Angular Velocity (3D only)
An angular velocity can be specified in addition to translational velocity in AUTODYN-3D.

The angular velocity is defined in radians / time and the axis of rotation is defined through the direction cosines of a vector passing through a point in space.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5





During the Fill of a Part, the angular velocity and spin axis are used to calculate the resultant x, y and z velocity at each node. These are then added to any specified translational velocities.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Initial Explicit Model Initialization

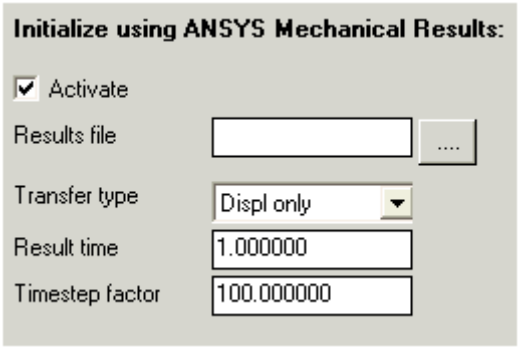
AUTODYN explicit analysis can be initialized (or pre-stressed) with the results from either a linear static structural, non-linear static structural or transient dynamic ANSYS Mechanical implicit analysis. Note that this feature requires the ANSYS results to be saved to a .rst file.

It is assumed that the same mesh is always used in both the implicit and explicit analyses; otherwise results will not be mapped onto the explicit mesh correctly. Note that quadratic elements may be used in the implicit model. In this case results for the corner nodes and average results for the element will be applied to the explicit mesh.

The user is advised to check the autodyn.log file shortly after starting the AUTODYN analysis as any errors occurring during the implicit results transfer will be echoed to this file.

Activate

Initialization of an explicit analysis under implicit analysis results can be selected using the Activate option under Initialize using ANSYS Mechanical results.



The menu shown above is used to configure the transfer of data from the ANSYS Mechanical to the AUTODYN solution and in which manner AUTODYN will be initialized.

Results File

Select the ANSYS .rst results file to be used to initialize the AUTODYN analysis.

Transfer Type

Either the Displacement Only or the Material State option as described below.

Transfer Type options are available to define what analysis results need to be transferred from the ANSYS implicit analysis to initialize the AUTODYN analysis:

1. Displacement Only

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Nodal displacements from an ANSYS linear static solution are used to initialize the AUTODYN nodal positions. These displacements are converted to constant nodal velocities and applied for a pre-defined time in order to obtain the required displaced co-ordinates. During this time elements stress/strains are calculated as normal by AUTODYN. Once the displaced nodal positions are achieved all nodal velocities are set to 0 and the solution is completely initialized.

- This option is applicable to unstructured solids (hexahedral and tetrahedral), shells and beams
- This option is only applicable to results from a linear, static structural analysis.
- Linear elastic materials only can be used in the explicit analysis when using the Displacement Only Option.
- Note that after AUTODYN model initialization static damping (Controls>Damping Options>Static Damping) needs to be applied to change the AUTODYN calculation from a dynamic solution to a state of stress equilibrium. For optimal convergence, the value chosen for the damping constant, R, may be defined by:

  R=2*timestep/T

  where timestep is the expected average value of the timestep and T is the longest period of vibration for the system being analyzed.

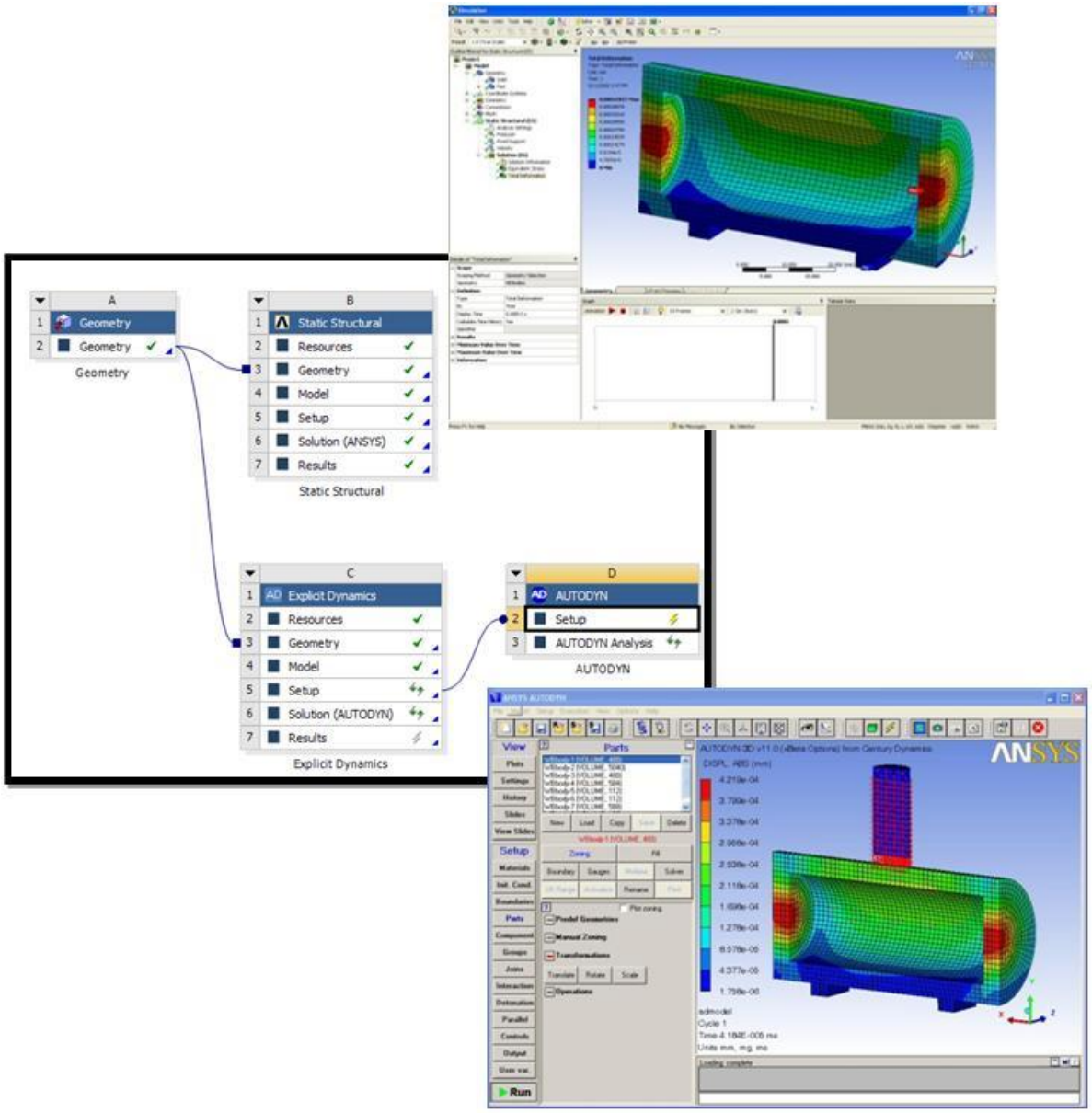
2. Material State

Nodal displacements and element stresses, strains, plastic strains and velocities from an ANSYS solution are used to initialize an AUTODYN analysis at cycle 0.

The AUTODYN solution is therefore completely initialized at the start of the analysis.

- This option is only applicable to unstructured solid elements (hexahedra and tetrahedral)
- This option is applicable to results from a linear static structural, non-linear static structural or transient dynamic Mechanical system.
- The following materials can be used in the AUTODYN solution with the Material State Option. Equations of State (linear, hyperelastic). Strength Models (elastic, von-Mises Johnson-Cook, piecewise, Zerilli Armstrong, Steinberg Guinan, Cowper Symonds, hyperelastic, bilinear isotropic hardening).
- Note that static damping can also be applied by using a static damping constant, but this may not be required.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Results Time

The result time from the ANSYS Mechanical implicit analysis at which to extract results from the .rst file. For a linear static solution this will always be 1.0. For a nonlinear or transient dynamic solution this time will correspond to a result time from the ANSYS Mechanical analysis.

Timestep factor

(Note that this option is only applicable for the Displacement Only method.) The initial timestep from the AUTODYN solution is multiplied by the timestep factor. The resulting time (PVELTM) is used together with the nodal displacements from the ANSYS Mechanical analysis to calculate constant nodal velocities. These nodal velocities are applied to the AUTODYN model over PVELTM in order to initialize the AUTODYN nodes to the correct positions.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Initial Conditions - Delete

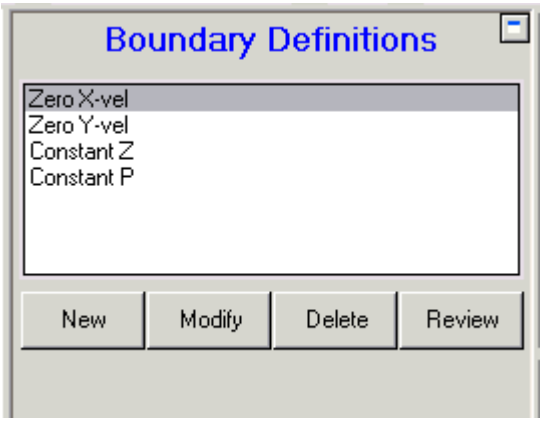
This window lets you delete Initial Condition Sets from your model.

Select the Initial Condition Set(s) you want to delete and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Boundaries

Boundaries



This panel lets you create **Boundary Conditions** that can be applied to **Parts**.

Boundary Condition List

The box at the top of this panel lists the Boundary Conditions you have currently defined for your model. You can select a Boundary Condition in this list.

New

Press this button to define a new Boundary Condition.

Modify

Press this button to modify the selected Boundary Condition.

Delete

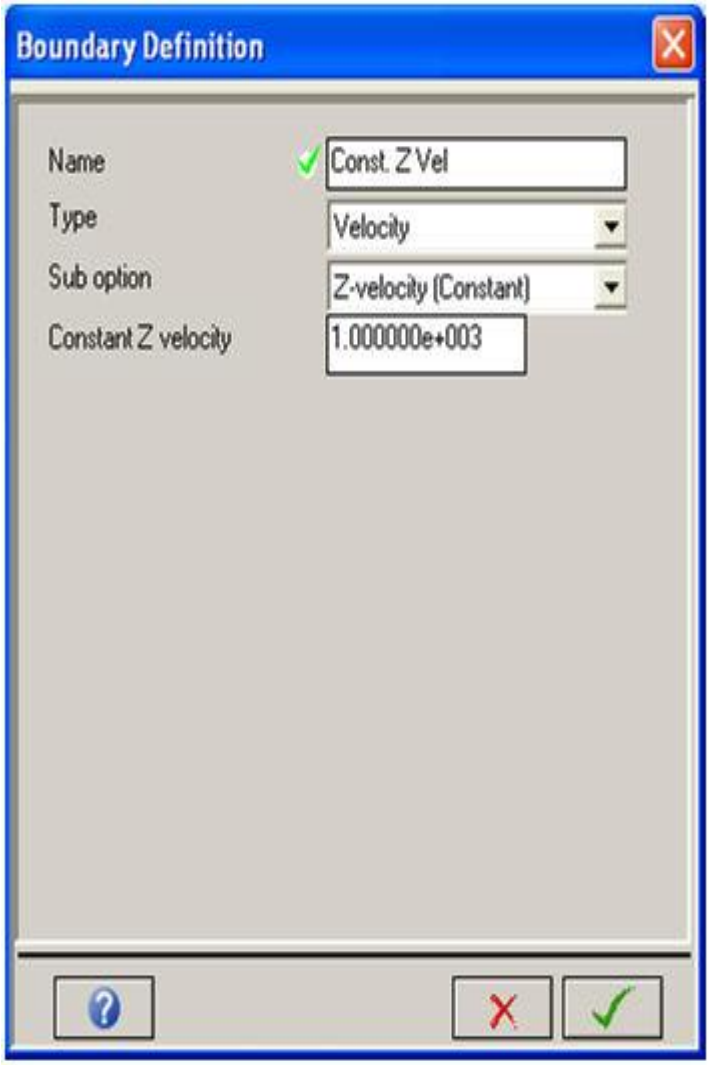
Press this button to delete one or more Boundary Conditions from your model.

Review

Press this button to review the parameters for all Boundary Conditions in a browser window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Boundaries - New / Modify



This window lets you define a new boundary condition.

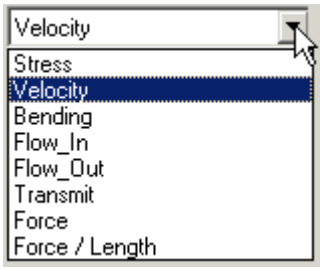
Name

Enter your name for the boundary condition in this field.

Type

Select the type of boundary condition you want to use from this pull-down menu :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Sub Option

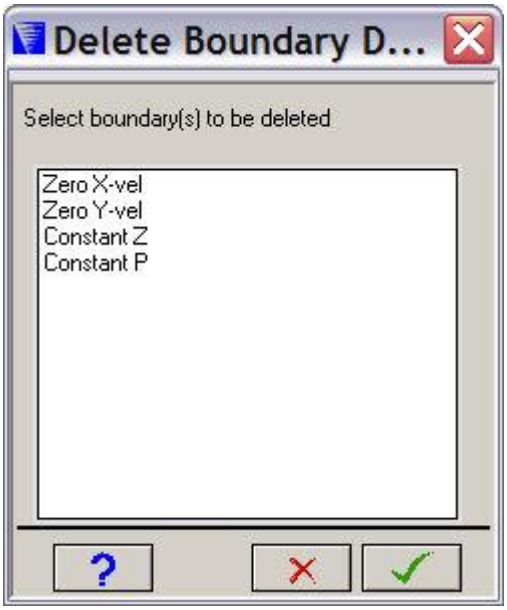
Select the sub-option you want to use from this pull-down menu :

Parameters

Enter values for any parameters associated with your selection in the fields provided (e.g. Constant Z Velocity).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Boundaries - Delete



This window lets you delete Boundary Conditions from your model.

Select the Boundary Condition(s) you want to delete and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

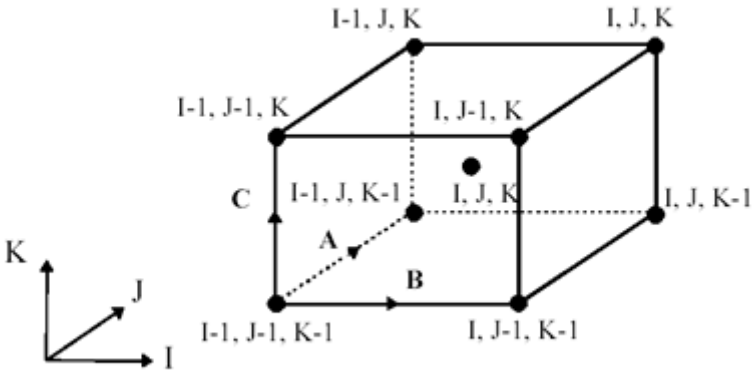
Parts

Parts

In AUTODYN, you build your model in Parts. There are two concepts for node, element and part data storage and processing available.

Structured Parts

For structured parts in AUTODYN, all information relating to the mesh and the elements in the mesh is stored and processed using the concept of I, J and K lines (IJK-space). This means that for an element whose quantities are stored at position (I, J, K), the nodes belonging to that element are automatically defined in IJK space as (I-1, J,K), (I, J-1,K), (I-1, J-1,K),.....(see the figure below).



Typical Connectivity for IJK-based Element in AUTODYN

The element connectivity is therefore implicitly defined by the structure within the I, J, K indexing. Information relating to each point in IJK space is also stored in this structure. Furthermore, in this structured approach, the node and element storage and processing is directly linked to the Parts in the model.

Each structured Part is a grid (or group of SPH nodes) that a particular solver operates on.

AUTODYN offers many tools for the generation of structured Parts as will be described below.

Unstructured Parts

For unstructured parts an alternative "Unstructured" approach to storing and processing information relating to the mesh is used for the solvers: The basic entities needed in the simulation are the Nodes, Elements, Faces Joints etc. In this scheme, these are stored in unique and independent global "Entity" lists, where an entity is a node, element, face, join etc. Connectivity between entities is determined by cross-referencing to the appropriate global entity list. The data associated with

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

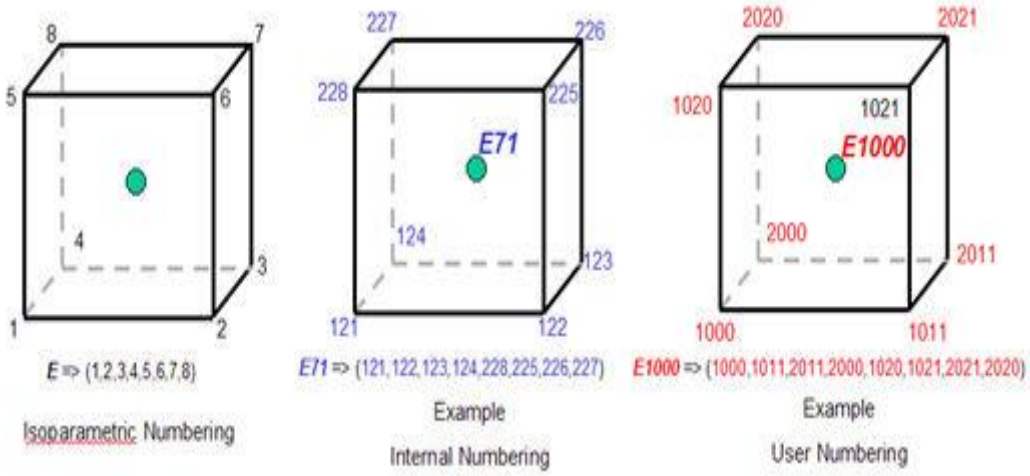
each entity is also stored in a manner most efficient for solving the equations and does not need to follow a set structure/order.

A global list of all elements relating to unstructured Parts is maintained. Each element is given a unique identifying number from one to total number of elements. Each element also has a user element number associated with it for the purposes of model manipulation and user identification.

Each element will have a list of variables associated with it the number and type of variables depending on the type of element (for example a hexahedral element containing an orthotropic material or a triangular shell with a rigid material).

Each element has three numbering systems for its connecting nodes (see figure)

- A local isoparametric number; used to loop over the nodes of an element in the correct order and solved the physical equations. Each element with the same topology has the same numbering system.
- An internal node number; the physical properties of each node are referenced via the global node lists. A connectivity table is used to cross-reference the local node number to an entry in the physical node list.
- A user node number; a number assigned by the user to define the element



Example Element and Node Numbering

For the unstructured solvers in AUTODYN, Parts are simply used to allow easy grouping and manipulation of various regions of the model. They are not related to the actual data storage for the elements and nodes, as in IJK-based Parts.

The default definition of a Part currently used for the unstructured solvers is: "A Part is a list of elements of the same classification (volume, shell, beam) that are connected together via the connectivity table of the elements"

Note that within each Part, all elements must be of the same class (volume, shell or beam), but not necessarily the topology. For example a shell Part may contain a mixture of quadrilateral and triangular elements. Nodes are only related to Parts through the element connectivity. Nodes can therefore exist in multiple Parts, for example if the same node is connected to a beam and shell element.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

AUTODYN doesn't offer any capability of generating unstructured elements and Parts manual. Unstructured elements and Parts can be generated in ANSYS Workbench and transferred to AUTODYN.

Furthermore unstructured models in MSC.Nastran or LS-Dyna format can also be imported directly into AUTODYN.

The following 3D Lagrangian unstructured element types are supported:

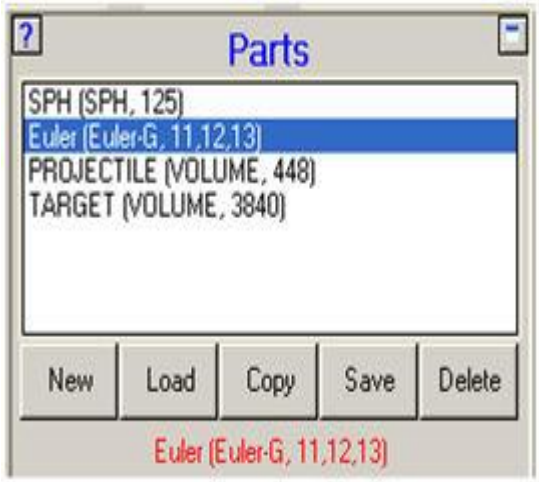
- *Volume elements*
  - *8 noded hexahedral elements*
  - *6 noded pentahedral elements (to be used as transition elements)*
  - *4 noded tetrahedral elements*
- *Shell elements*
  - *4 noded quad elements*
  - *3 noded tria elements (to be used as transition elements)*
- *Beam elements*
  - *2 noded beam elements*

The following 2D Lagrangian unstructured element types are supported:

- *Volume elements*
  - *4 noded quadrilateral volume elements*
  - *3 noded triangular volume elements*

Note:

1. The 2D Lagrangian unstructured element types have Beta status and therefore may not be as stable or robust as expected in a general release version.
2. A 2D or 3D analysis involving unstructured parts with only rigid materials may result in unphysical results, because rigid bodies do not have a minimum time step associated with them and therefore too large displacements may occur from cycle to cycle and contact can be missed. For this type of analysis the user needs to manually define a small time step for the analysis.



This panel lets you create structured Parts and modify the structured and unstructured **Parts** of your model.

Part List

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The box at the top of this panel lists the Parts you have currently defined for your model. You can select a Part in this list.

The size of the structured parts is displayed in terms of maximum I, J and K of the part (see Part Euler above). Structured SPH parts are displayed in terms of the total number of SPH nodes, because SPH parts don't use elements.

For unstructured parts the size of part is displayed as the total number of elements (see Parts PROJECTILE and TARGET above).

New
Press this button to create a new structured part.

Load
Press this button to load a structured Part from a Part Library.

This option is not available for unstructured Parts because for these types of Parts the ANSYS Workbench project data storage can be used as a part library.

Copy
Press this button to copy an existing Part to a new Part. This is valid for both structured as well as unstructured Parts.

Save
Press this button to save structured Parts to a Part Library. This option is not available for unstructured Parts because for these types of Parts the ANSYS Workbench project data storage can be used as a part library.

Delete
Press this button to delete a structured/unstructured Part.

The remaining dialog in this panel depends on the Solver used by the selected Part.

All Structured Solvers except SPH

SPH Solver

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Unstructured Solvers

Zoning
Press this button to zone a structured Part.

Unstructured Parts cannot be generated; an unstructured Part can only be translated, rotated or scaled.

Fill
Press this button to fill a part with materials and initial conditions.

Geometry (Zoning)
Press this button to define geometric objects for **SPH** Parts

Pack (Fill)
Press this button to pack geometric objects with **SPH** nodes.

Boundary
Press this button to apply boundary conditions to a Part.

Gauges
Press this button to assign gauges to a part.

Motions
Press this button to assign motion constraints to a structured **ALE** Part.

Solver
Press this button to select solver options for a Part.

IJK Range
Press this button to modify the IJK range for a structured Part

Activation
Press this button to set activation / deactivation times for a structured Part.

For unstructured Parts activation / deactivation time cannot be set (they are always activated).

Rename
Press this button to rename a Part.

Print

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

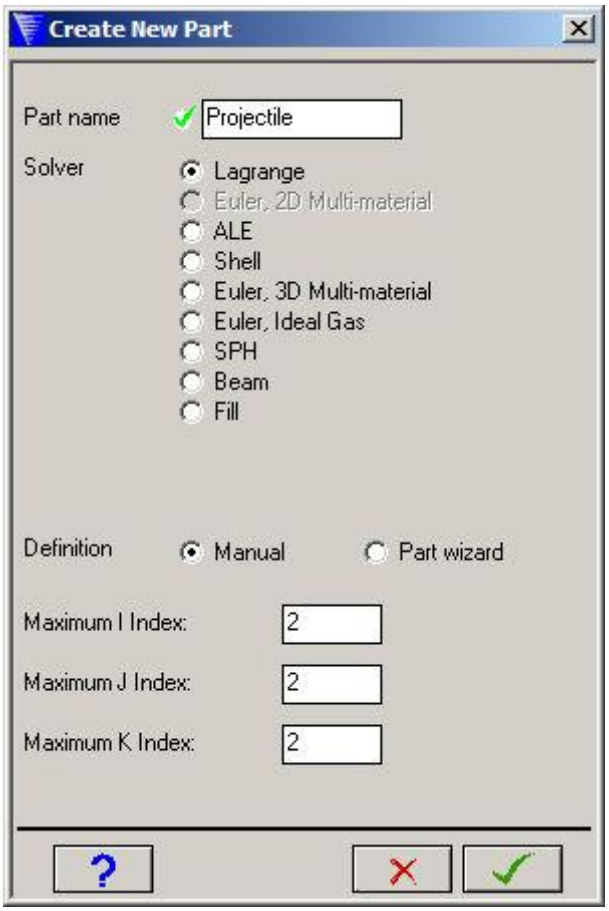
Press this button to set print ranges for a Part.

This option is not available for unstructured Parts.

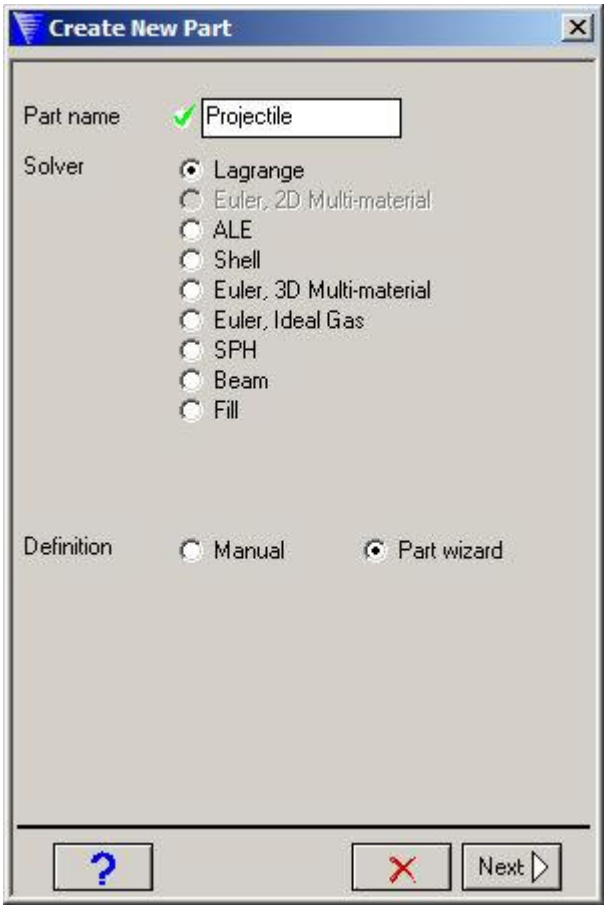
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

New

Parts - New



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



This window lets you create a new structured Part.

Part Name

Enter your name for the Part in this field.

Solver

Select the Solver you want to use for the Part.

Definition

You can select to define your Part manually or using the Part Wizard (the Part Wizard cannot be used for SPH or Beam Solvers, so these options are not displayed for these solvers)

Manual

If you choose this option, fields appear for you to specify the maximum I, J and K range for your Part (left window above).

After pressing you return to the Parts panel where you can define your Part manually using the **Zoning** and **Fill** buttons.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Part Wizard

If you choose this option, you advance through three more dialog windows that guide you through the complete definition of your part, using one of a number of pre-defined geometries (**Predefs**).

The first dialog window lets you choose the Predef you want to use.

The second dialog window lets you define your zoning.

The third dialog window lets you fill your part.

Press

to advance through the dialog windows, then press in the last window to create your Part.

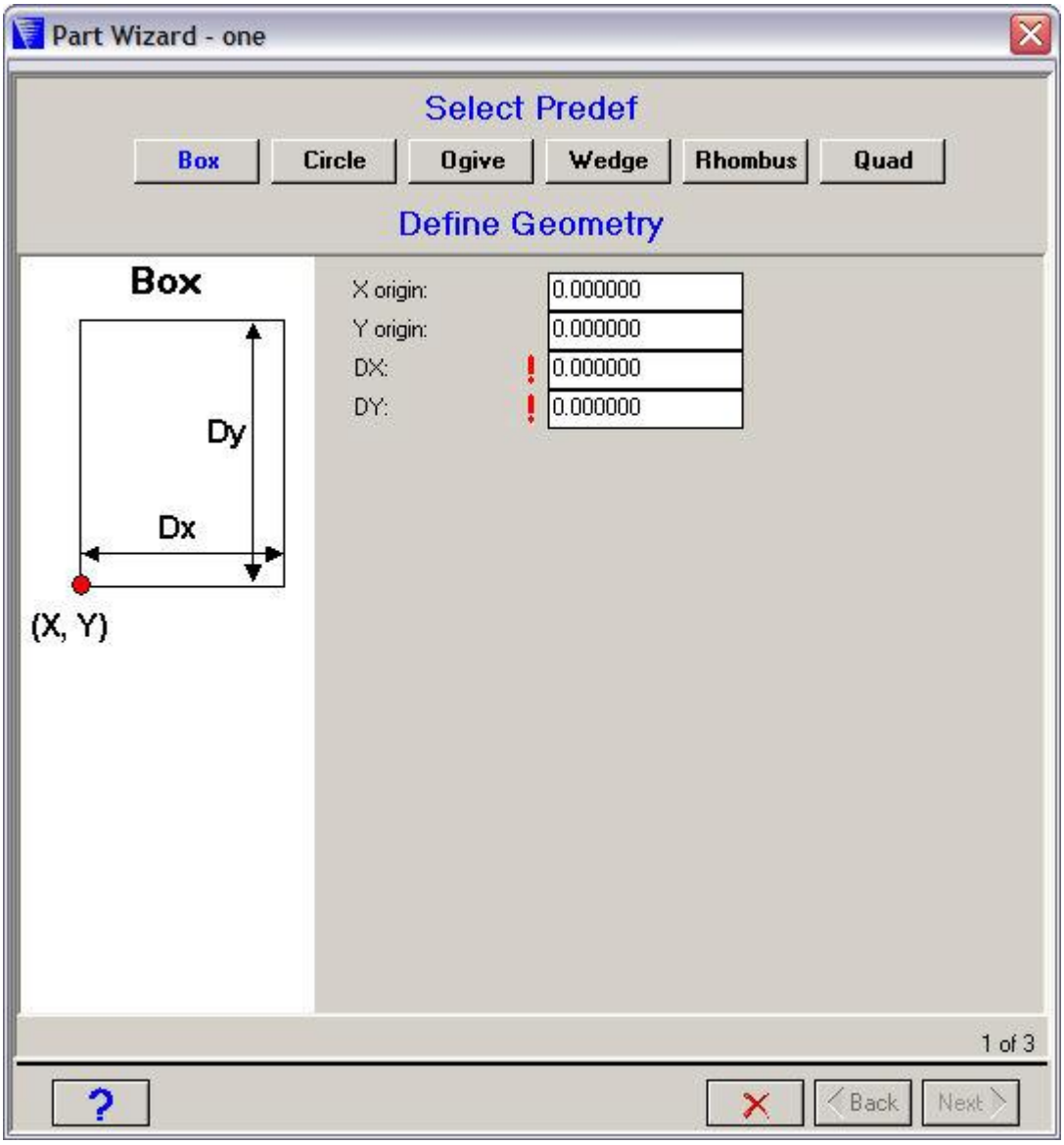
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Select Predef (2D)

This window lets you select the **Predef** you want to use for the 2D Part you are creating.

Press one of the five buttons at the top of this window to select your Predef :

Box



This predef is a box with its axes parallel to the X and Y axes.

Origin

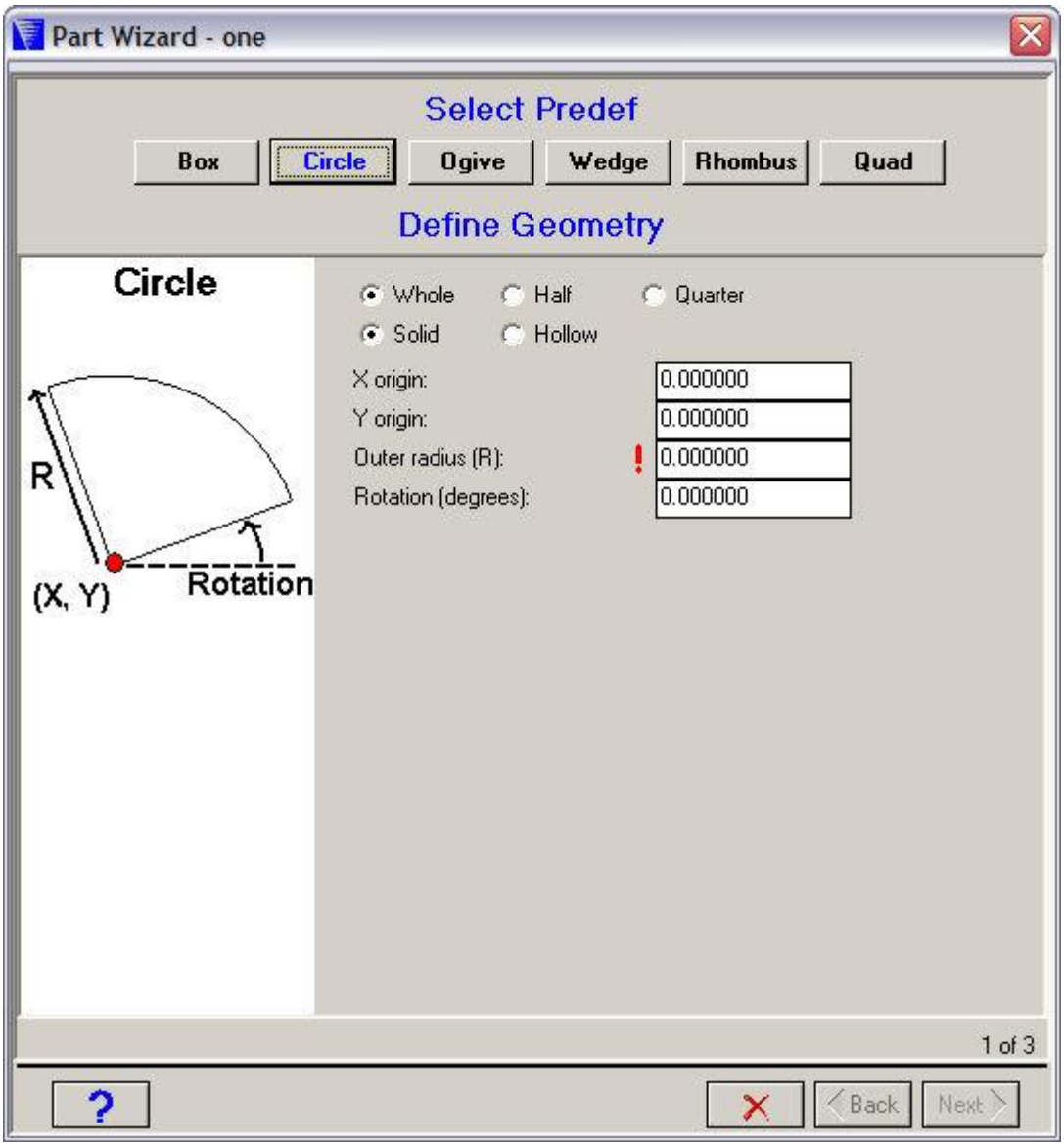
Enter the X and Y coordinates of the lower corner of the box in the fields provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Box Dimensions

Enter the dimensions of the box (DX, DY) in the fields provided

Circle



This predef is a circle.

Section

Choose the section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow cylinder.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

If you choose a hollow cylinder, you will have to specify inner and outer radii for the cylinder.

Origin

Enter the X and Y coordinates for the center of your circle in the fields provided.

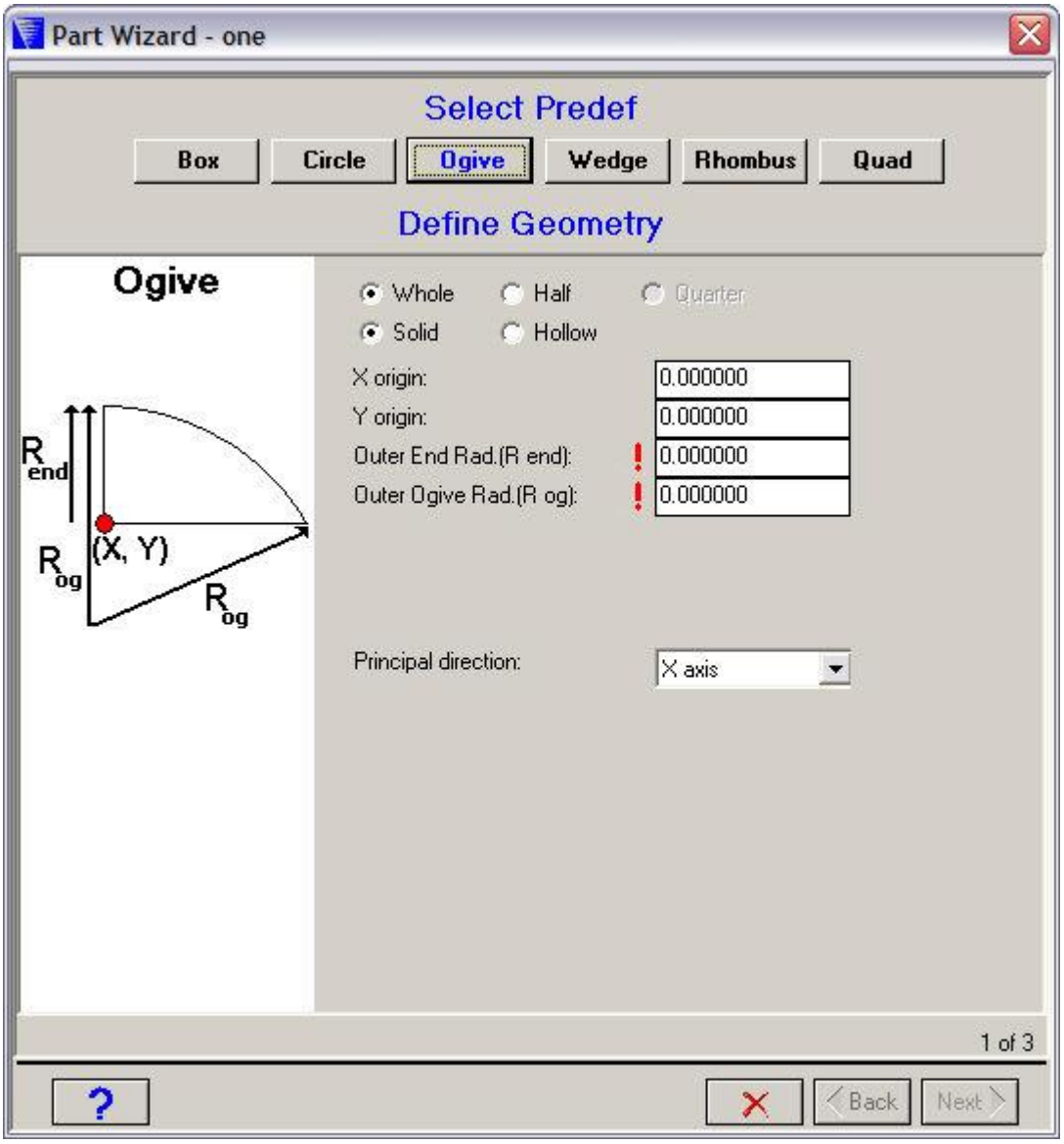
Outer Radius

Enter the outer radius of your circle.

Rotation

Enter the rotation (in degrees) of your circle relative to the X-axis.

Ogive



This predef is an ogive with its axes parallel to the X and Y axes.

Section

Choose the section you want to generate (Whole or Half). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow ogive.

If you choose a hollow ogive, you will have to specify inner and outer radii for the ogive.

Origin

Enter the X and Y coordinates for the center of your ogive in the fields provided.

End Radius

Enter the end radius of your ogive in this field.

Ogive Radius

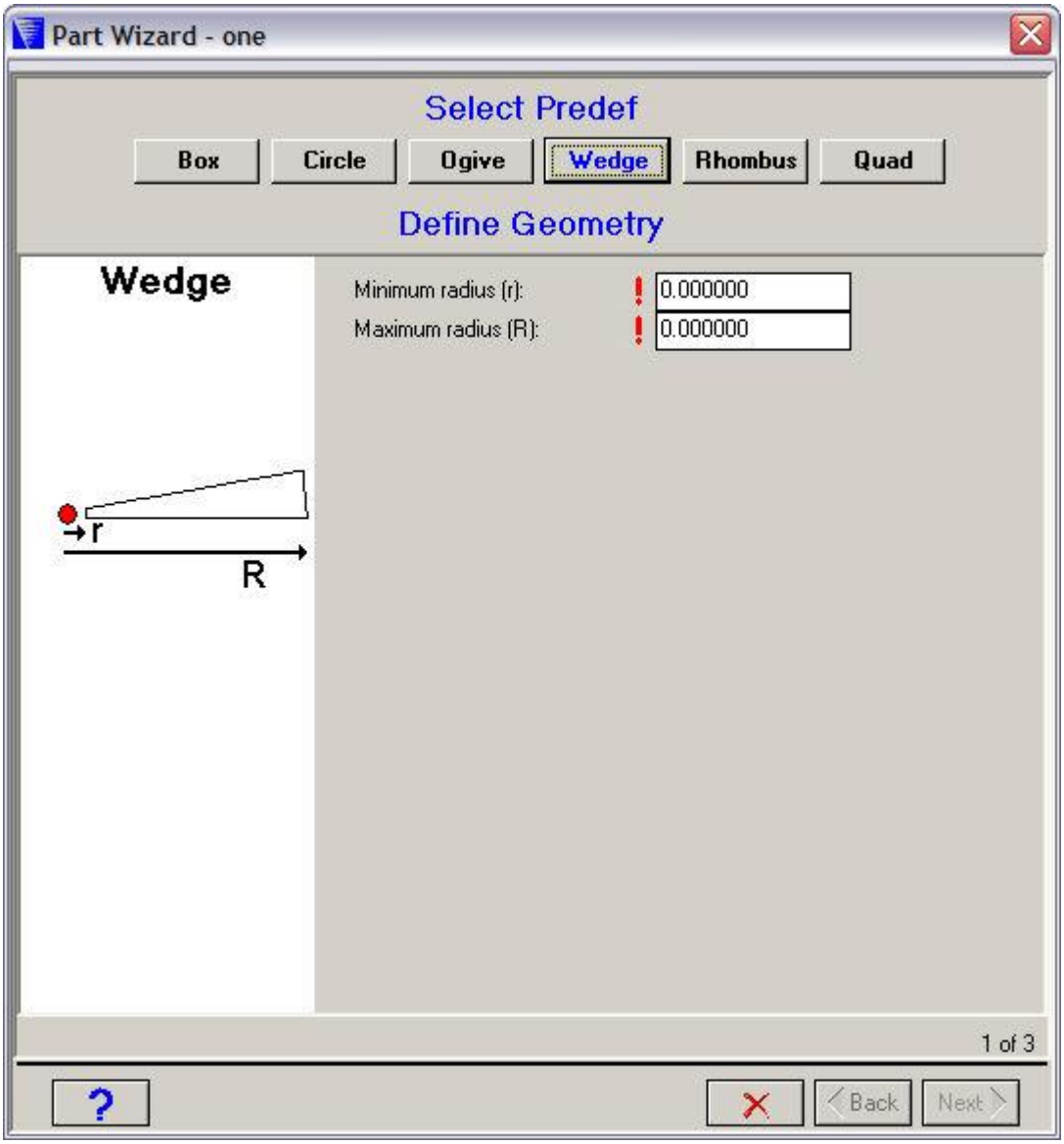
Enter your ogive radius in this field.

Principal Direction

Select the direction of the principal axis for your ogive from the pull-down menu

Wedge

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



This predef is a wedge with its lower edge lying on the X-axis. The wedge angle is automatically set by AUTODYN.

This predef is useful for running 1D cylindrical or spherical symmetric problems in 2D.

Minimum radius

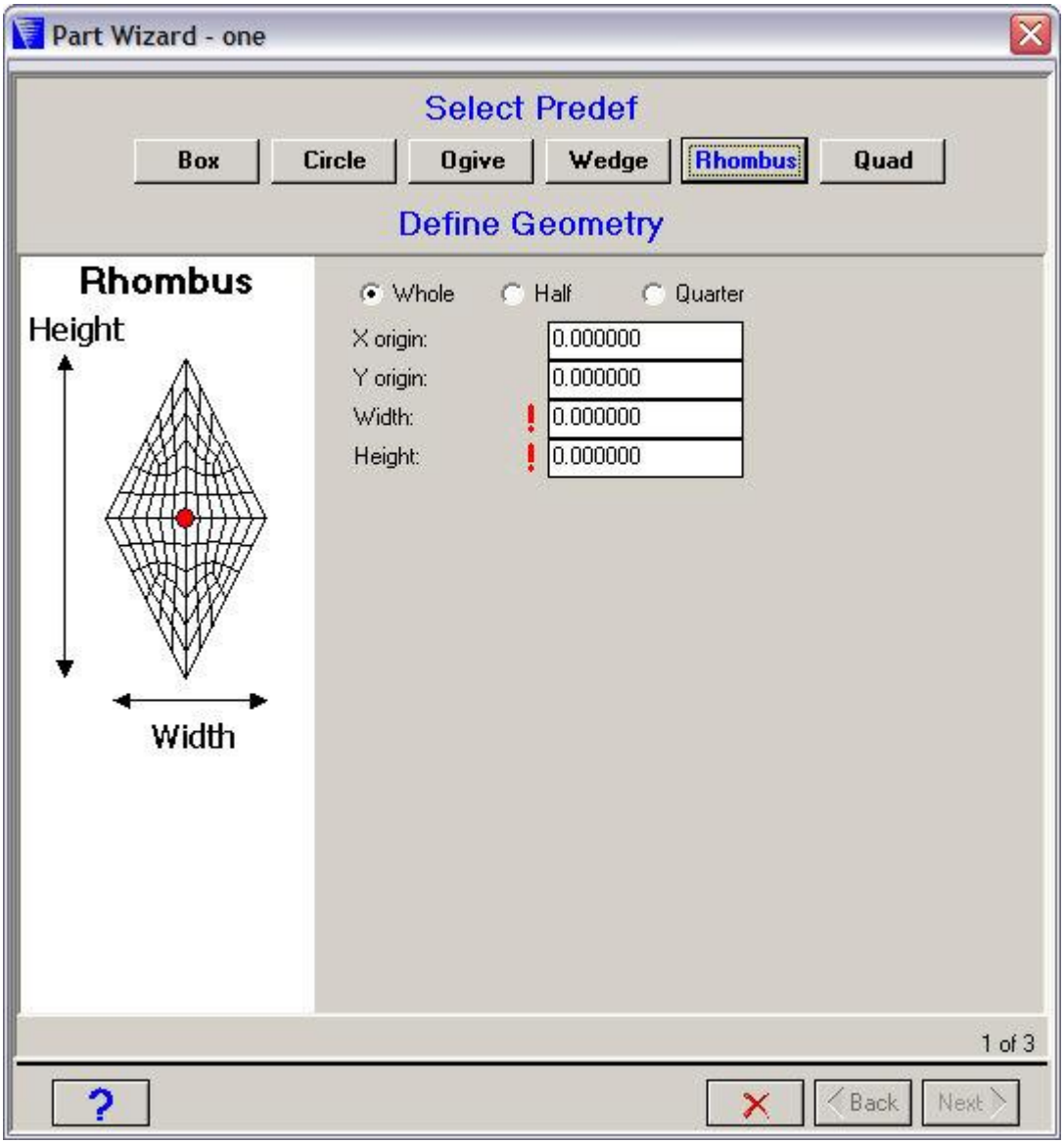
Enter the minimum radius of the wedge (this cannot be zero).

Maximum radius

Enter the maximum radius of the wedge.

Rhombus

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



This predef is a rhombus with its diagonals parallel to the X and Y axes.

Section

Choose the section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Origin

Enter the X and Y coordinates for the center of your ogive in the fields provided.

Width

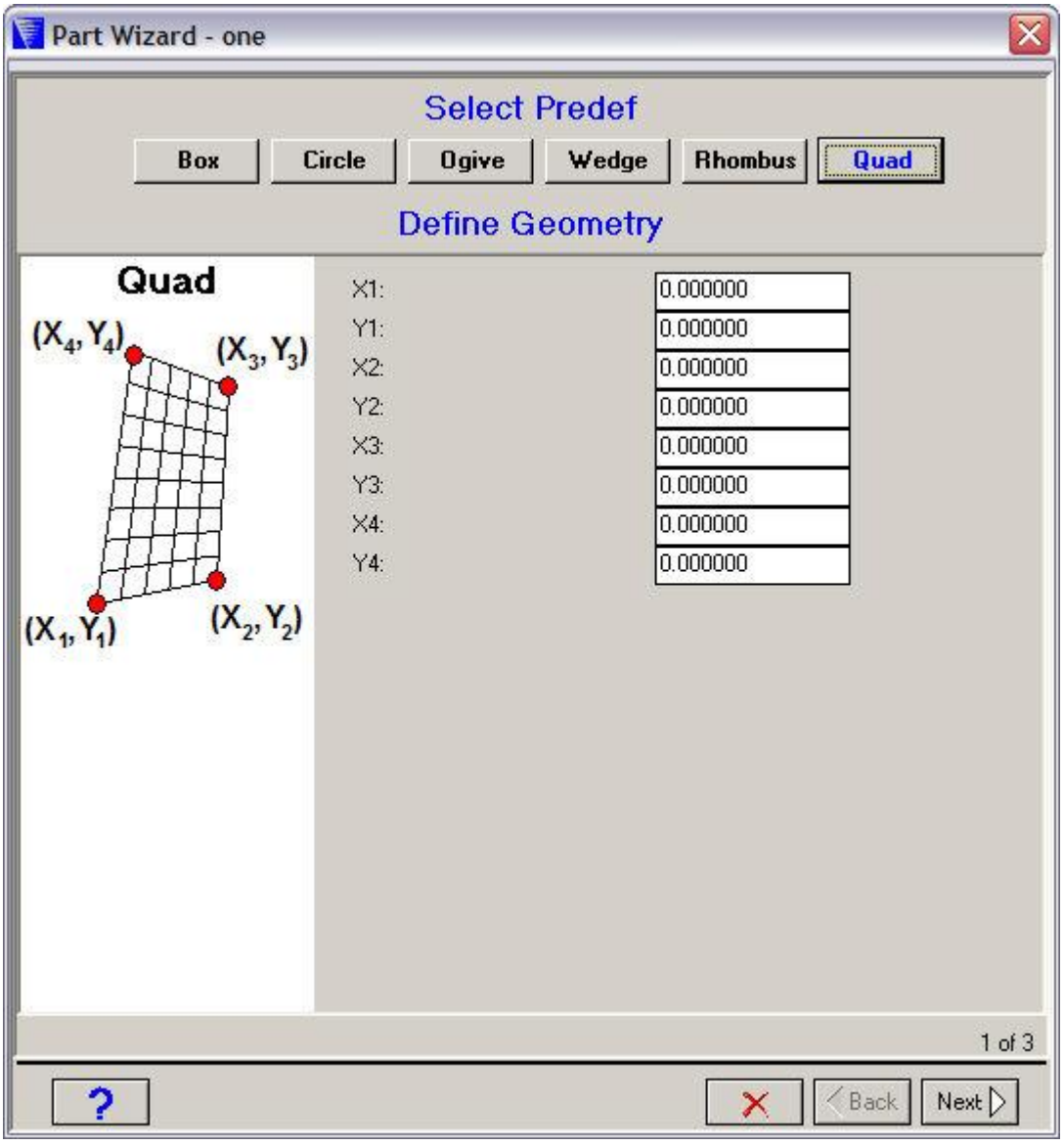
Enter the width of the rhombus.

Height

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the height of the rhombus.

Quad



Coordinates X1, Y1:

Enter coordinates of the first point of the quad.

Coordinates X2, Y2:

Enter coordinates of the second point of the quad.

Coordinates X3, Y3:

Enter coordinates of the third point of the quad.

Coordinates X4, Y4:

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter coordinates of the fourth point of the quad.

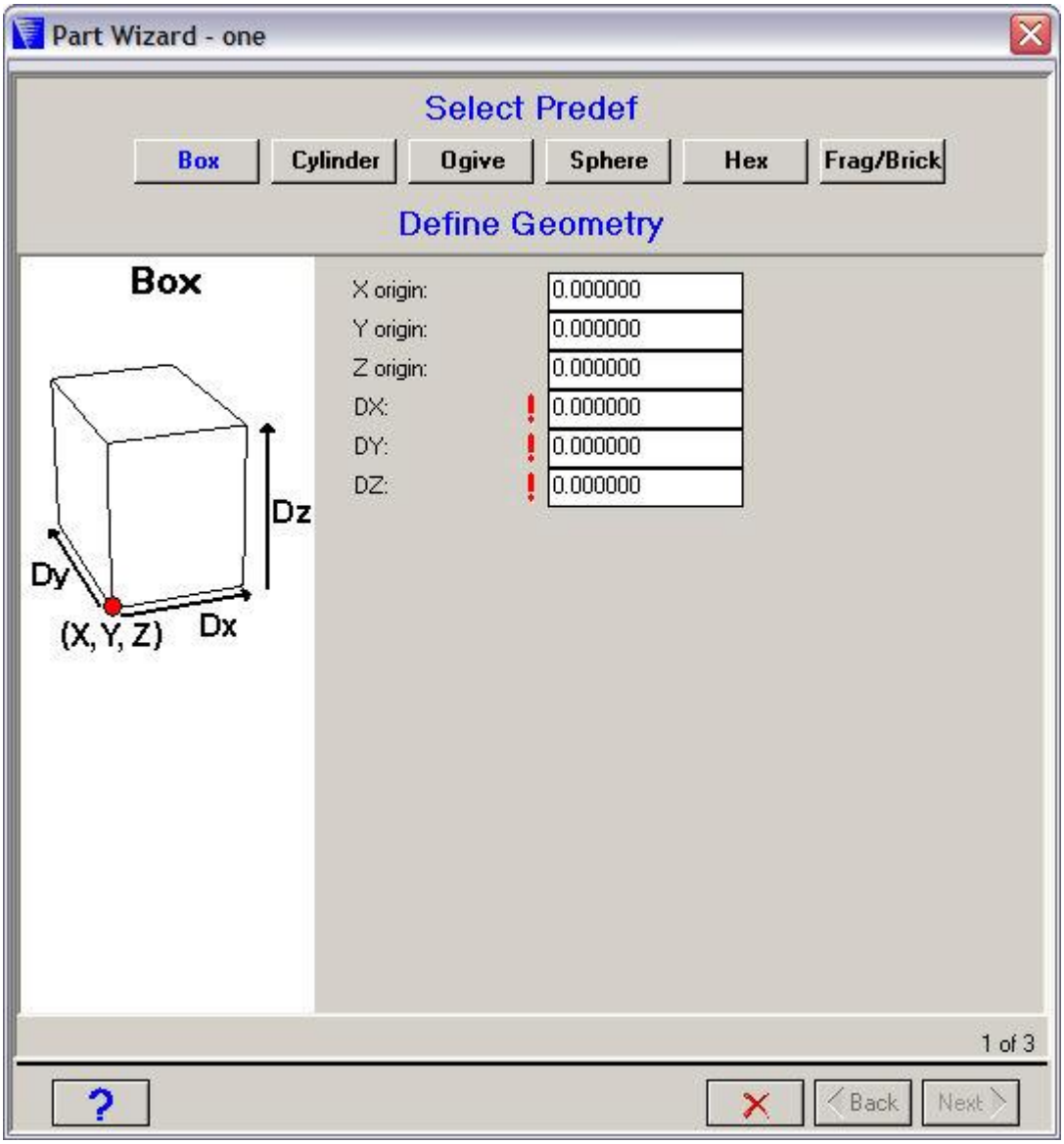
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Select Predef (3D)

This window lets you select the **Predef** you want to use for the 3D Part you are creating.

Press one of the six buttons at the top of this window to select your Predef :

Box



This predef is a box with its axes parallel to the X, Y and Z axes.

Origin

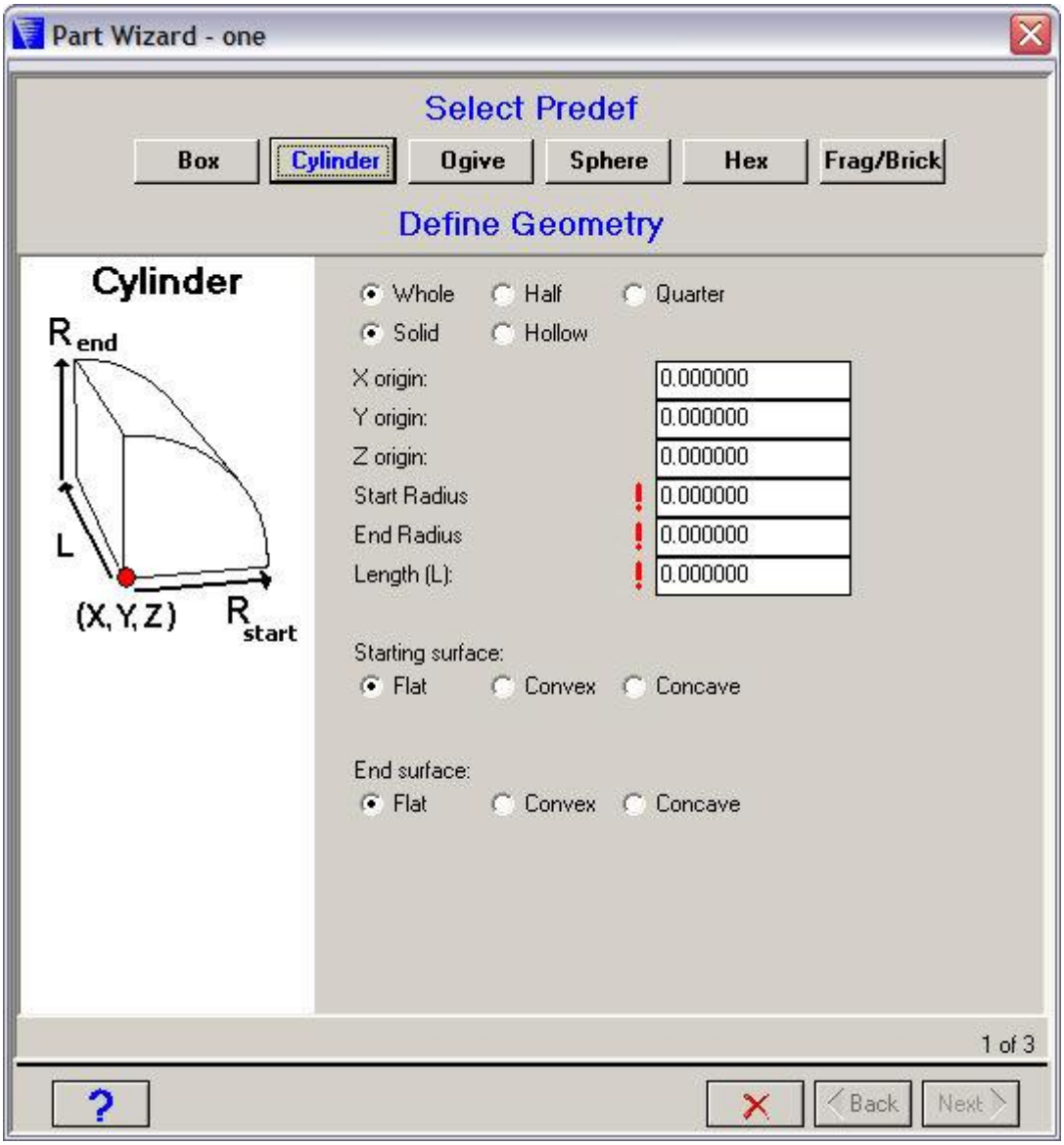
Enter the X, Y and Z coordinates of the lower corner of the box in the fields provided.

Box Dimensions

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the dimensions of the box (DX, DY, DZ) in the fields provided

Cylinder



This predef is a cylinder with its axis parallel to the Z axis.

Section

Choose the section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow cylinder.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

If you choose a hollow cylinder, you will have to specify inner and outer radii for the cylinder.

Origin

Enter the X, Y and Z coordinates for the center of the starting surface of your cylinder in the fields provided.

Start Radius / End Radius

You can specify different radii for the starting and end faces of your cylinder. For a regular cylinder (constant radius), set the starting radius equal to the end radius.

Length

Enter the length of your cylinder in this field.

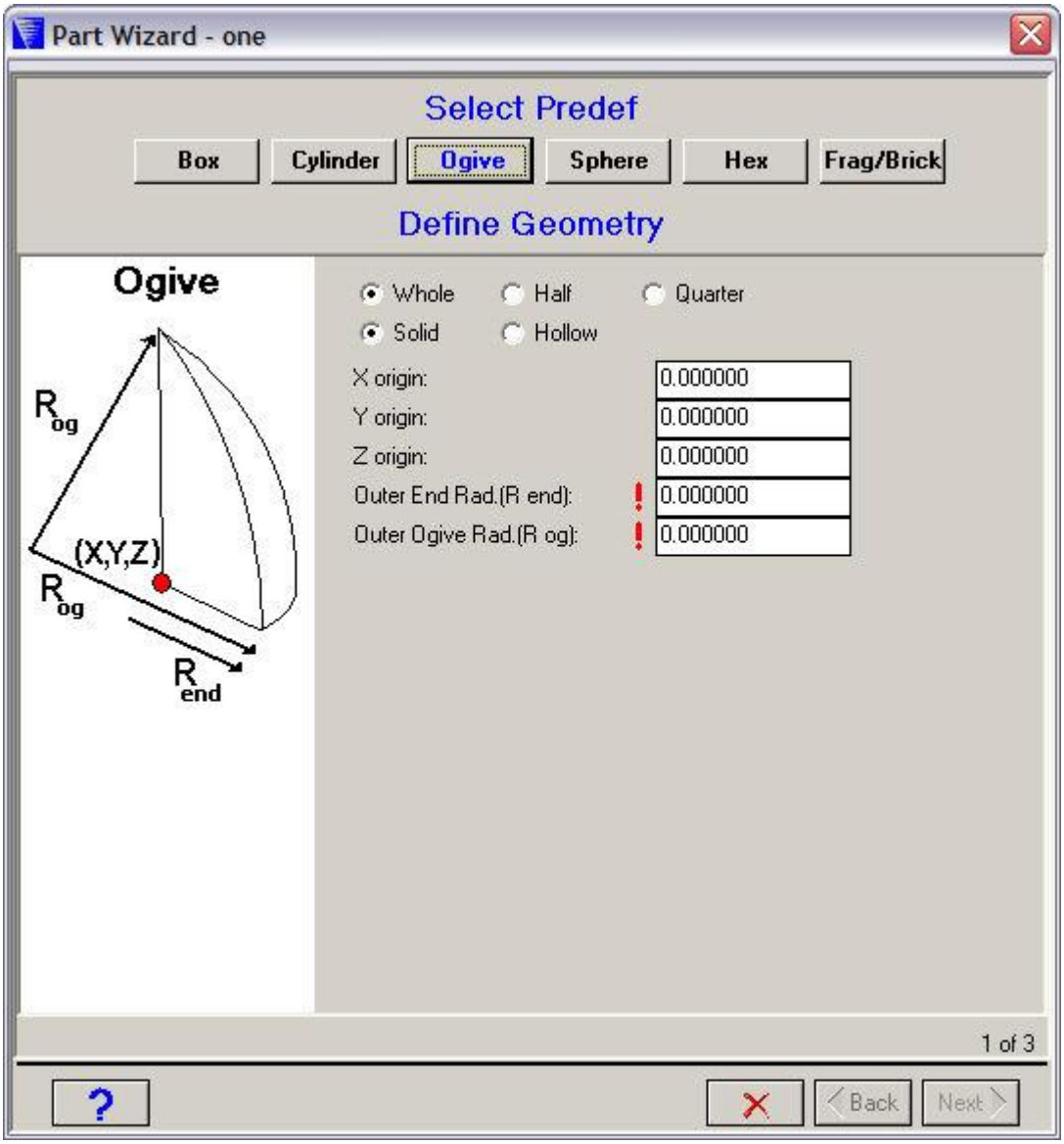
Start Surface / End surface

The starting and end surfaces of your cylinder can be flat, concave or convex.

If you select a concave or convex surface, you will have to specify its radius of curvature.

Ogive

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



This predef is an ogive with its axis parallel to the Z axis.

Section

Choose the section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow ogive.

If you choose a hollow ogive, you will have to specify inner and outer radii for the ogive.

Origin

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the X, Y and Z coordinates for the center of your ogive in the fields provided.

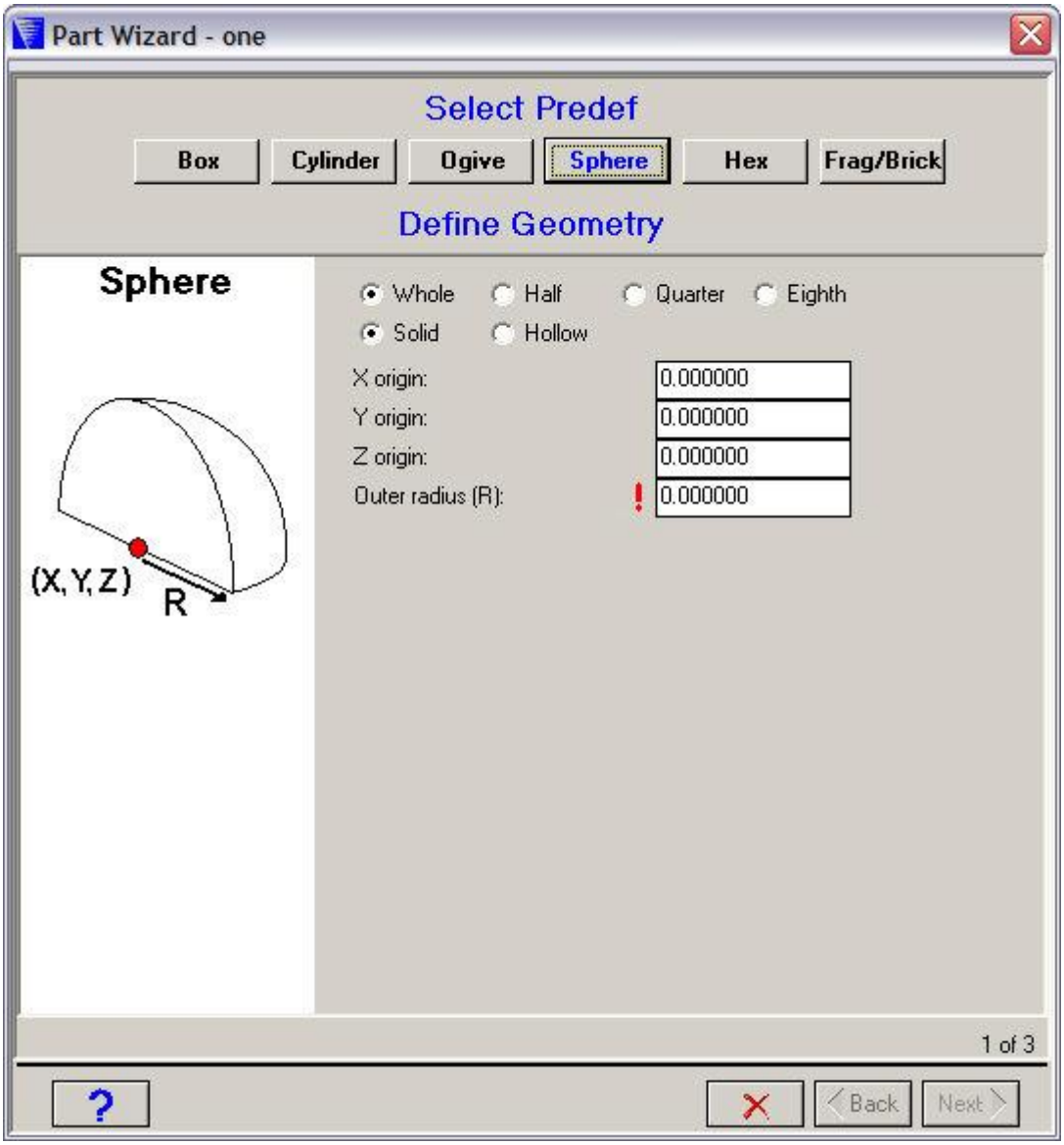
End Radius

Enter the end radius of your ogive in this field.

Ogive Radius

Enter your ogive radius in this field.

Sphere



This predef is a sphere.

Section

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Choose the section you want to generate (Whole, Half, Quarter or Eighth). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow sphere.

If you choose a hollow sphere, you will have to specify inner and outer radii for the sphere.

Origin

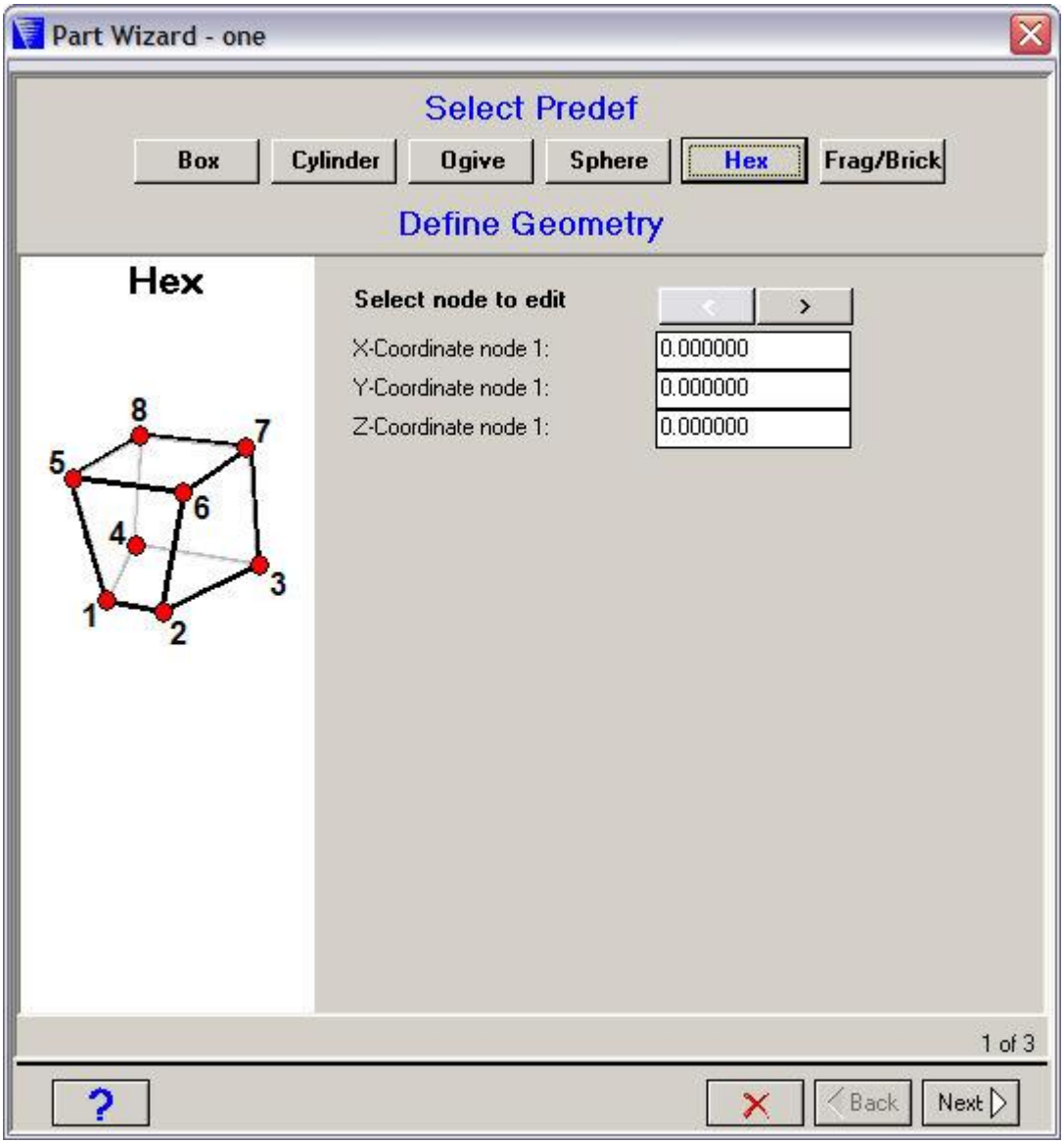
Enter the X, Y and Z coordinates for the center of the sphere in the fields provided.

Radius

Enter the radius of the sphere in this field.

Hex

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Select Node to Edit

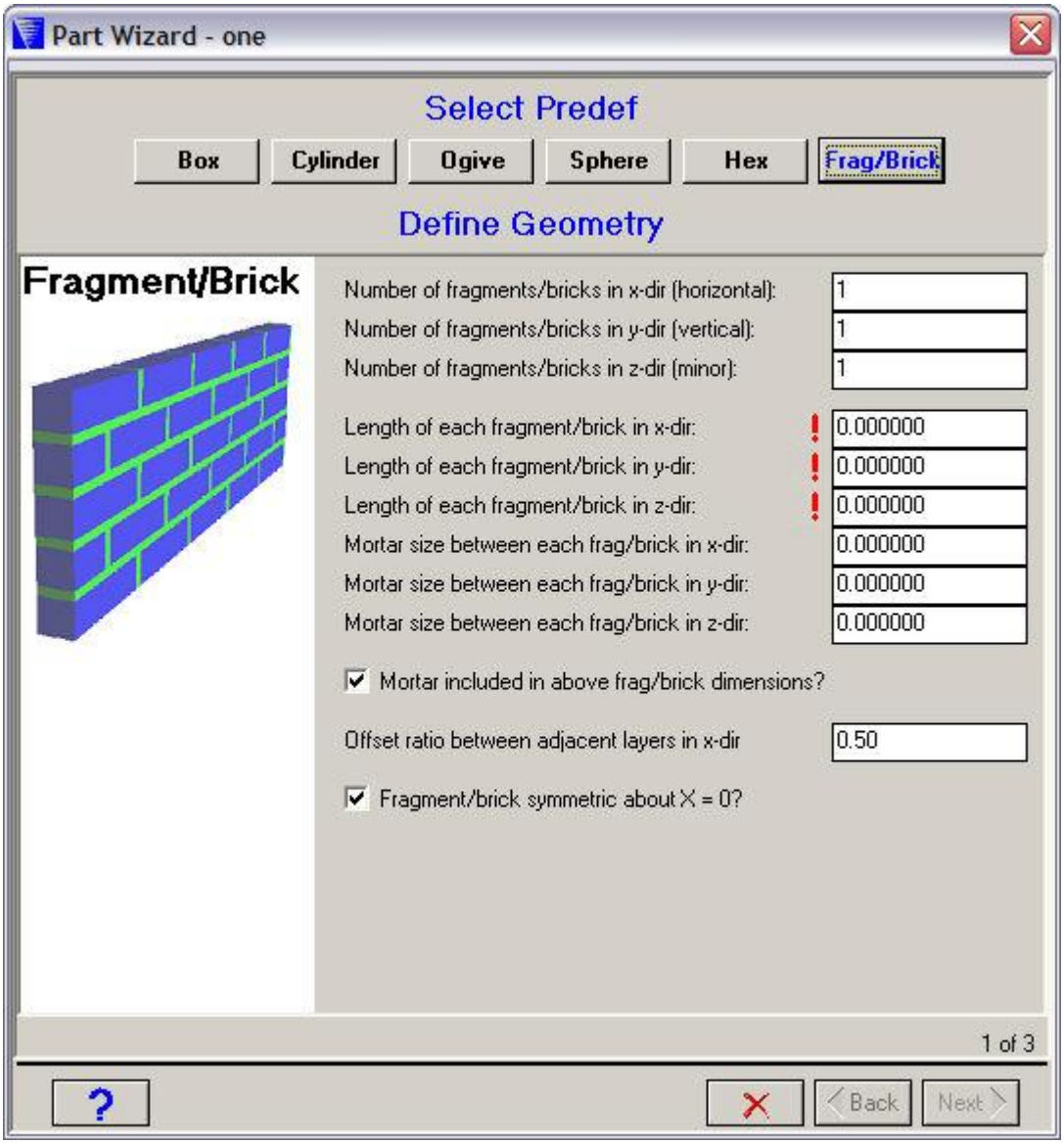
Use the arrow buttons to select each of the 8 nodes in turn.

Coordinates

Enter the X, Y and Z coordinates of the node.

Frag/Brick

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



The fragment/brick wall is built with its axes parallel to the X, Y and Z axes.
The X direction runs the length of the wall.
The Y direction runs the height of the wall.
The Z direction runs through the thickness of the wall.

Number of fragments/bricks

Use these fields to set the number of fragments/bricks in the X, Y and Z directions.

Length of each fragment/brick

Use these fields to set the length of a single brick/fragment in the X, Y and Z directions.

Mortar size between each fragment/brick

Use these fields to set the mortar size (or gap) between fragments/bricks.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Mortar included in dimension?

Check this box if you want the mortar size to be included in the length of each brick/fragment.

Offset ratio

Set the offset ratio for the fragments/bricks in this field.
This is how the bricks are staggered along the length of the wall.
e.g. if the offset is 0.5 (the default), the bricks will be staggered halfway across each other as shown in the above figure.

Fragment/brick symmetric about X=0?

Check this box if you want the wall to be symmetric about X=0.

Enter the required dimensions for your predef and press

to advance to the next dialog window (**Define Zoning**).

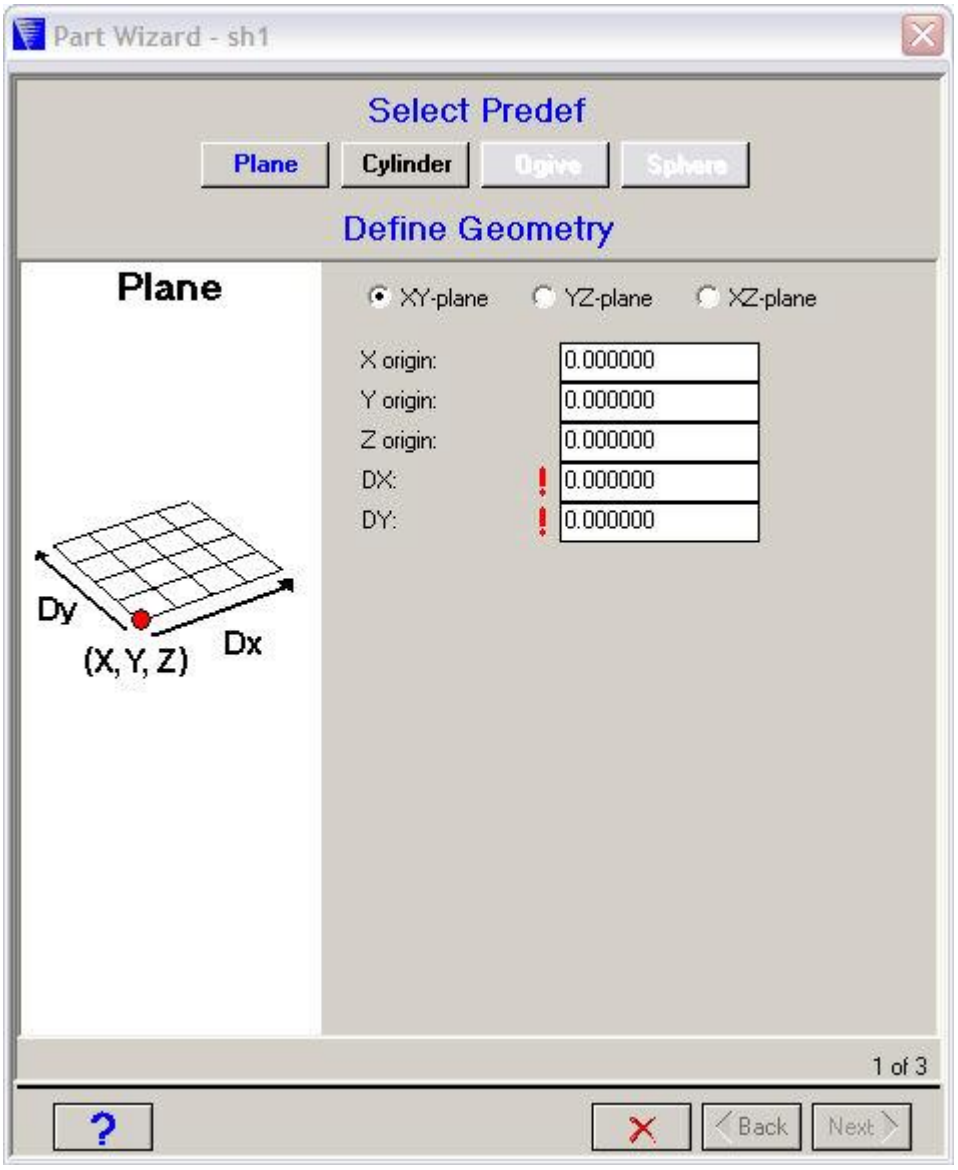
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Select Predef (3D Shell)

This window lets you select the **Predef** you want to use for the Shell Part you are creating.

Press one of the two buttons at the top of this window to select your Predef :

Plane



This predef is a plane perpendicular to the X Y or Z axis.

Plane Orientation

Select the orientation of the plane (e.g. **XY-plane** is a plane perpendicular to the Z axis).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

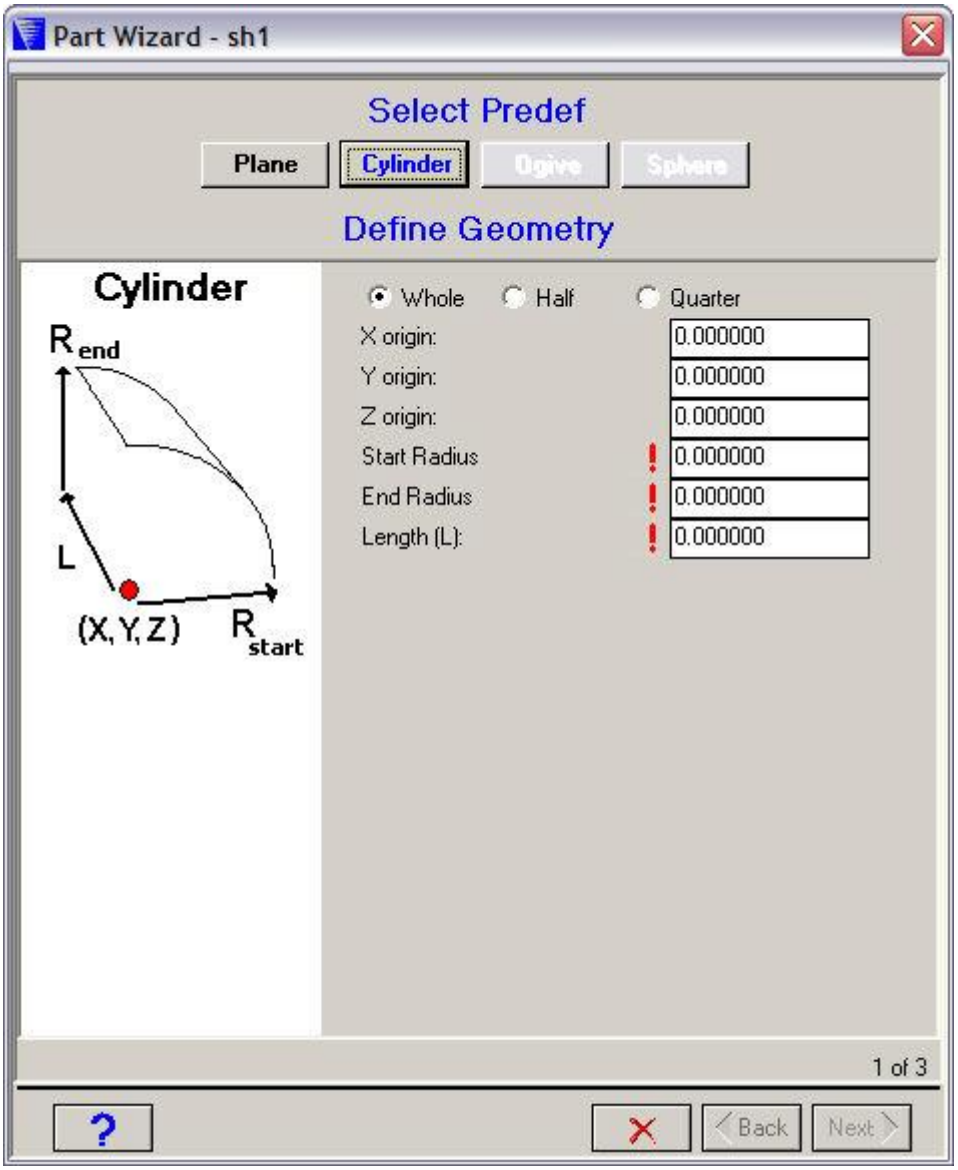
Origin

Enter the X, Y and Z coordinates of the lower corner of the plane in the fields provided.

Box Dimensions

Enter the dimensions of the box (DX, DY, DZ) in the fields provided

Cylinder



This predef is a cylinder with its axis parallel to the Z axis.

Section

Choose the section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Origin

Enter the X, Y and Z coordinates for the center of the starting surface of your cylinder in the fields provided.

Start Radius / End Radius

You can specify different radii for the starting and end faces of your cylinder. For a regular cylinder (constant radius), set the starting radius equal to the end radius.

Length

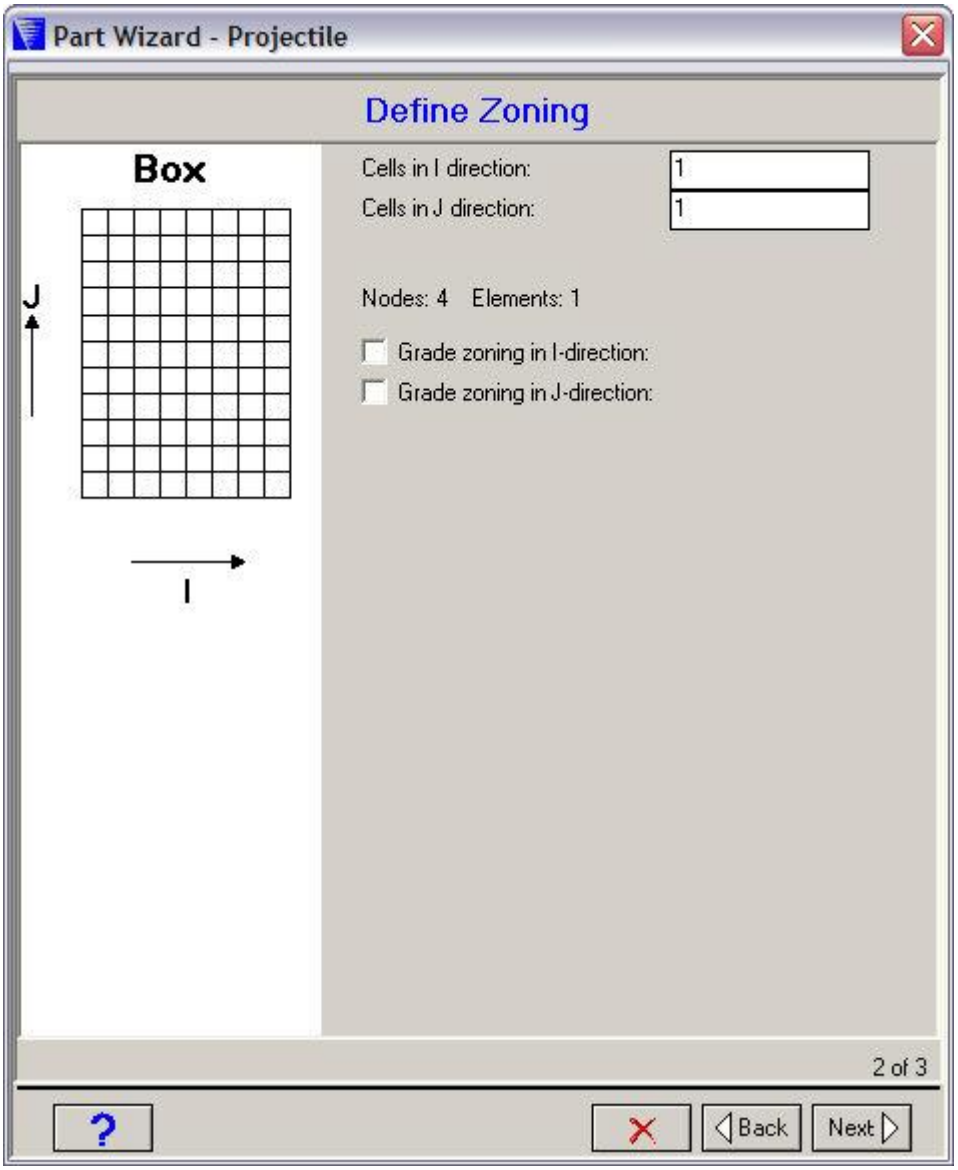
Enter the length of your cylinder in this field.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Define Zoning (2D)

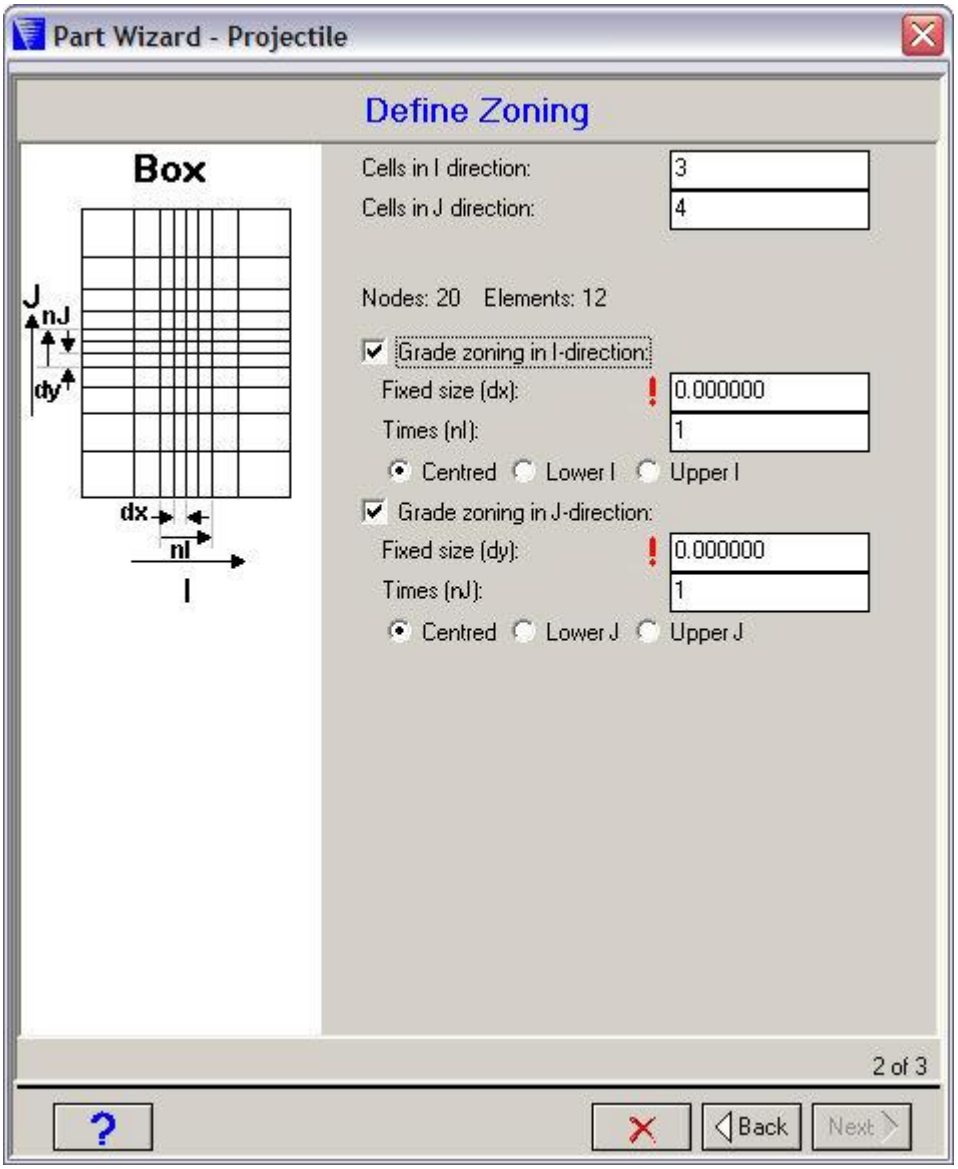
This window lets you Define the zoning for your 2D Part.

Box



No Grading

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Grading in X and Y

Cells

Enter the number of cells (elements) you want to use in the I and J directions in fields provided. The corresponding number of nodes and elements required for your part will be displayed below.

Grade zoning

Check the boxes if you want to grade your zoning in any direction.

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

Fixed size

Enter the dimension you want to use for the fixed size cells in this field.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Times

Enter the number of fixed size cells you want to use in this field.

Position

Specify where you want to place the fixed size cells relative to the full I/J/K range.

Centered - at the center of the full I/J range
Lower - at the lower end of the I/J range
Upper - at the upper end of the I/J range

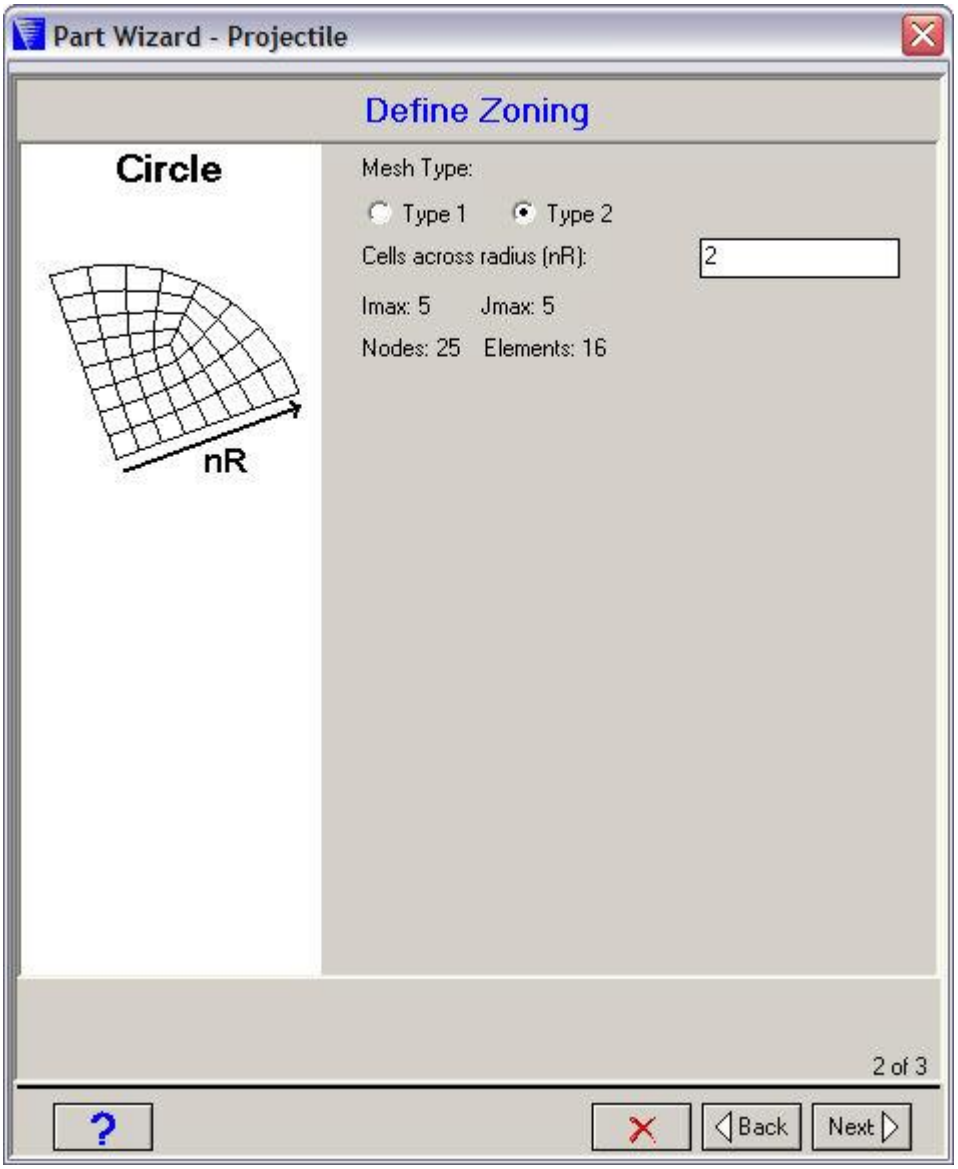
e.g. ten fixed size cells centered :

e.g. twelve fixed size cells at the lower end :

e.g. eight fixed size cells at the upper end :

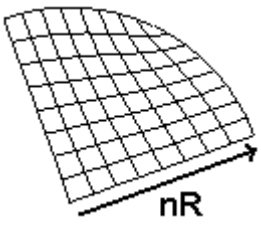
Circle

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

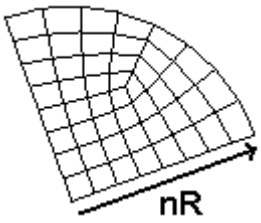


Mesh Type

Select a Type 1 or Type 2 mesh for your circle.



Type 1



Type 2

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but about a quarter of the elements generated for this mesh are unused.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

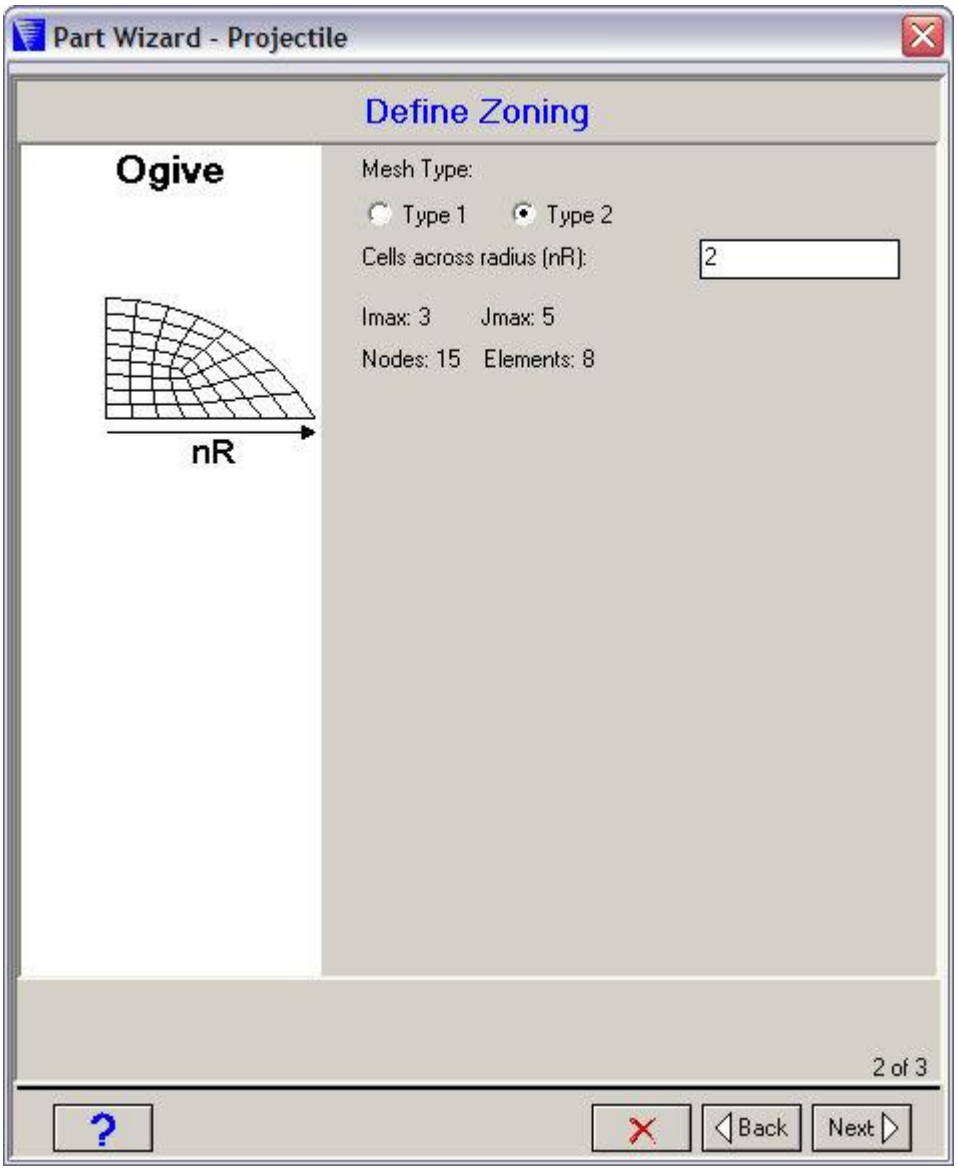
Cells across radius

Enter the number of cells (elements) you want to use across the radius of your circle in this field.

Number of elements

Displayed below the input fields is the index space that will be used for your Part and the number of nodes and elements required.

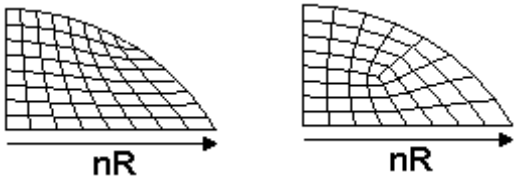
Ogive



Mesh Type

Select a Type 1 or Type 2 mesh for your ogive.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Type 1 **Type 2**

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but some of the elements generated for this mesh are unused.

Cells across radius

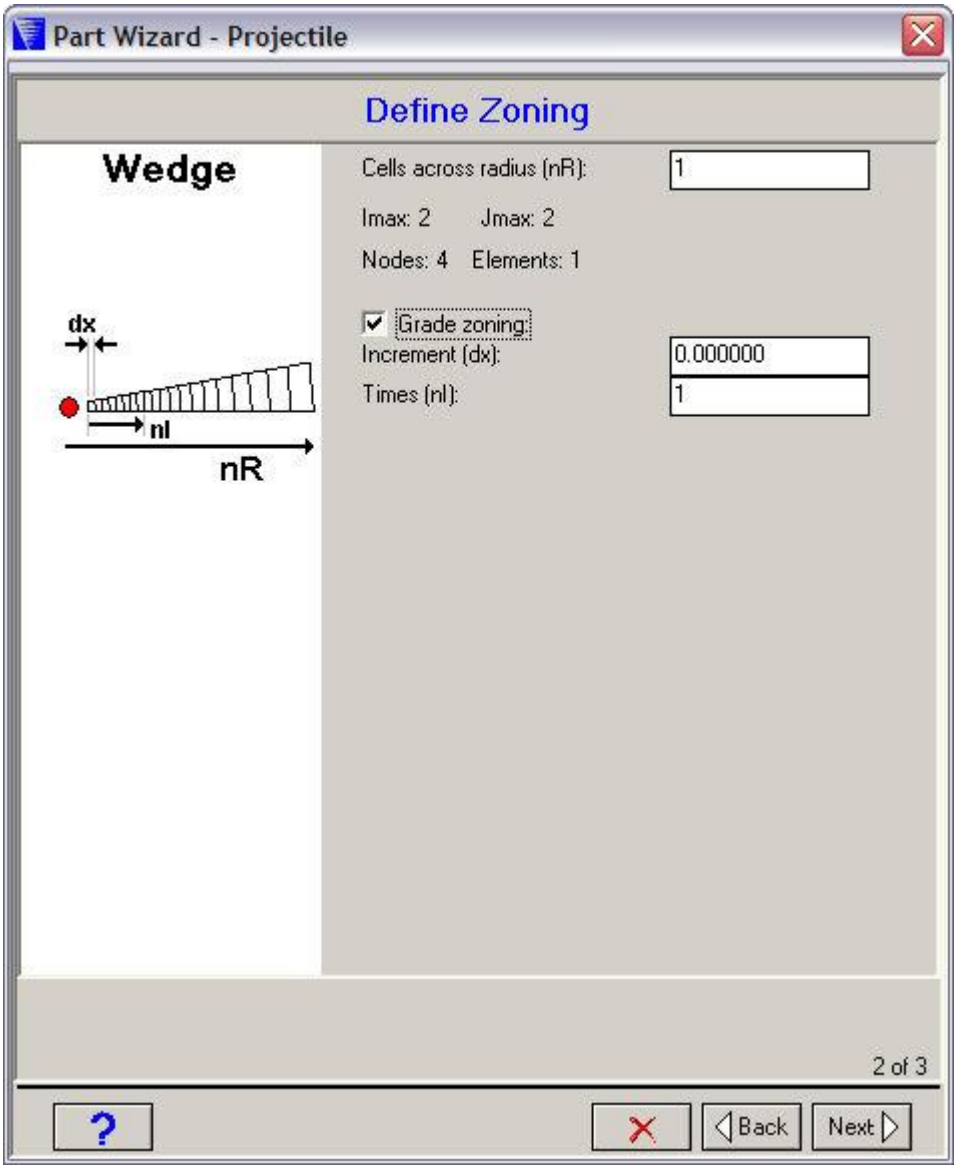
Enter the number of cells (elements) you want to use across the radius of your ogive in this field.

Number of elements

Displayed below the input fields is the index space that will be used for your Part and the number of nodes and elements required.

Wedge

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Cells across radius

Enter the number of cells (elements) you want to use across the radius of your wedge in this field.

Number of elements

Displayed below the first input field is the index space that will be used for your Part and the number of nodes and elements required.

Grade zoning

Check this box if you want to grade your zoning.

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

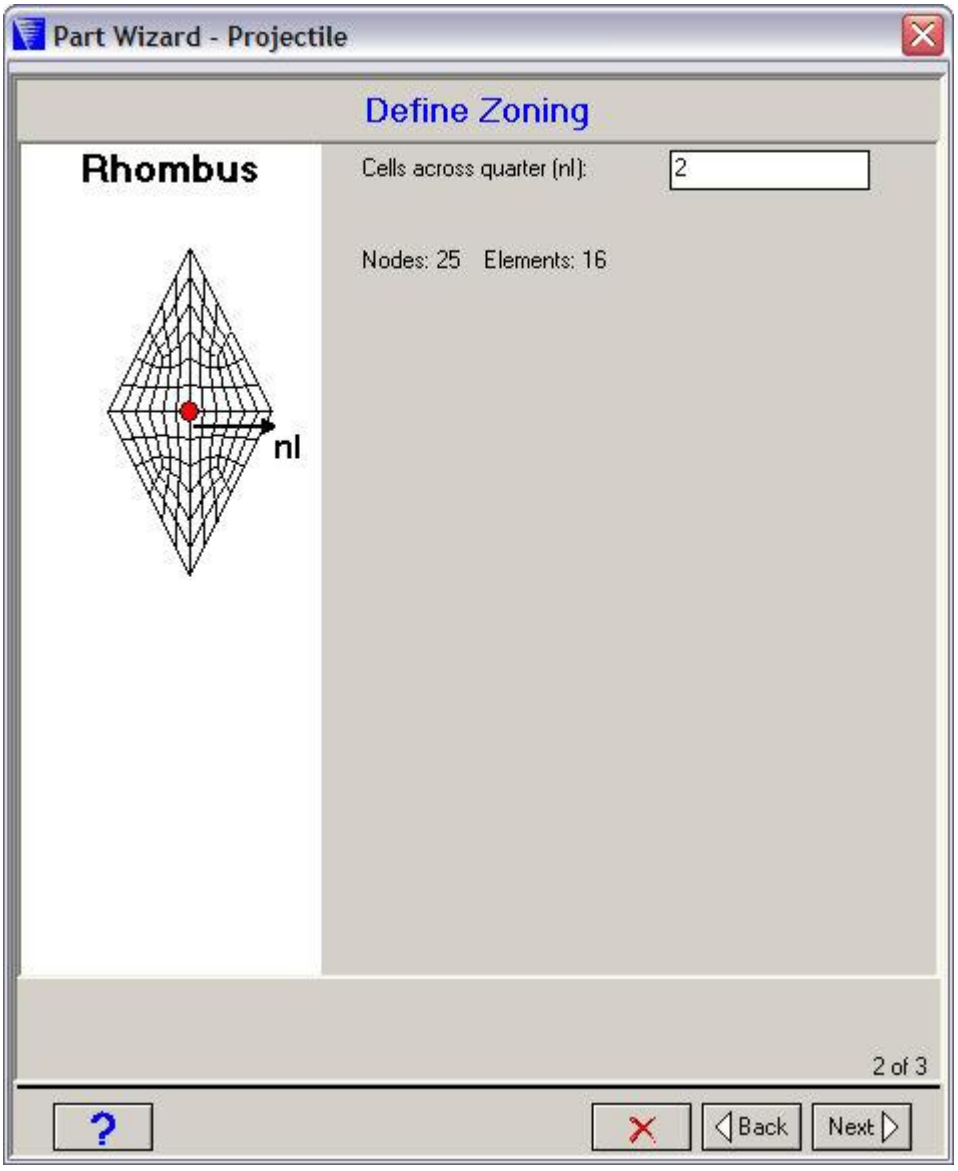
Increment

Enter the dimension you want to use for the fixed size cells in this field.

Times

Enter the number of fixed size cells you want to use in this field.

Rhombus



Cells across quarter

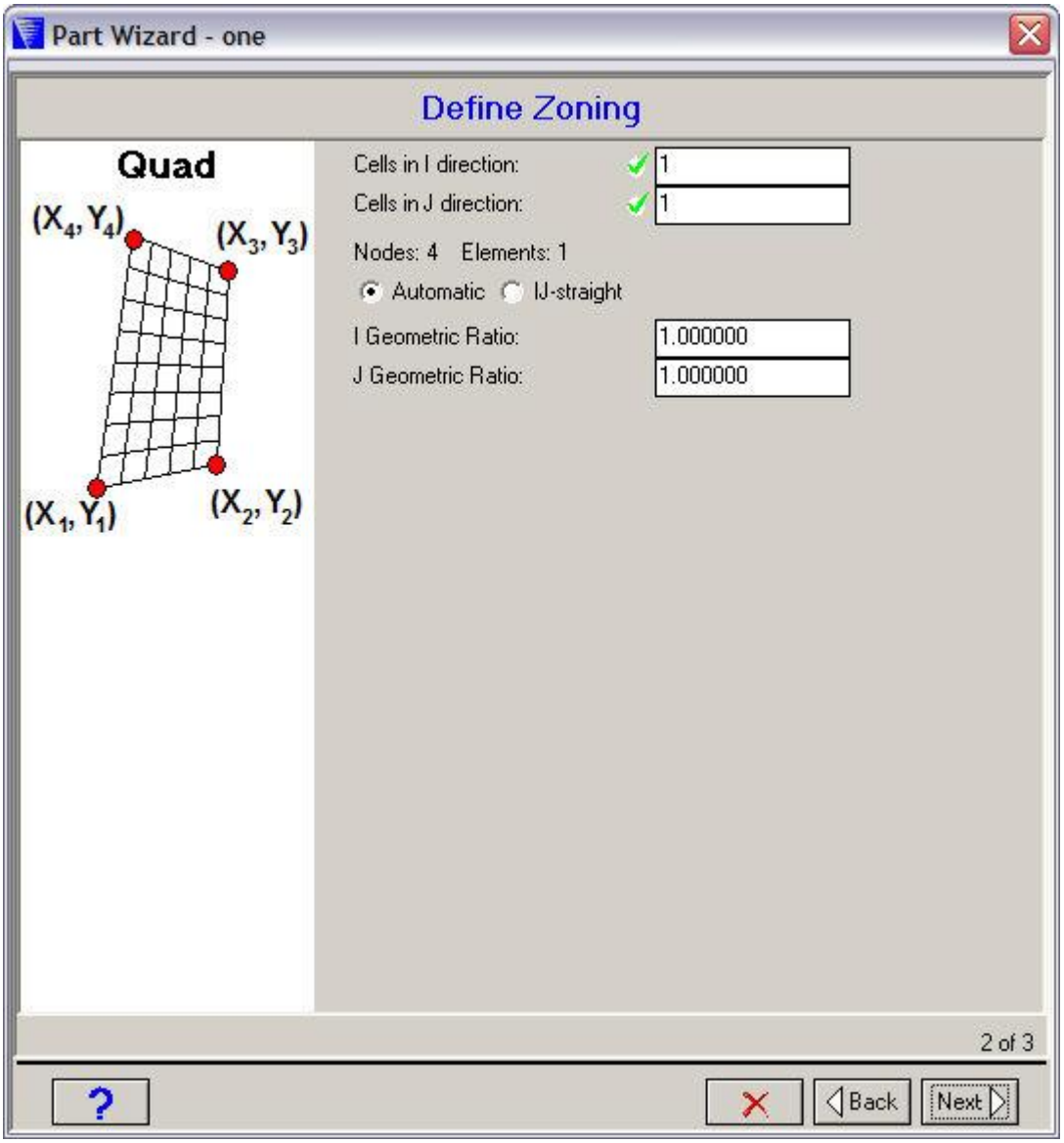
Enter the number of cells (elements) you want to generate across a quarter of the rhombus.

Number of elements

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Displayed below the input field is the index space that will be used for your Part and the number of nodes and elements required.

Quad



Cells in I/J direction

Enter the number of cells (elements) you want to use in the I and J directions in fields provided. The corresponding number of nodes and elements required for your part will be displayed below.

Automatic

Select this option if you want the internal nodes to be spaced equipotentially.

IJ-Straight

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select this option if you want the nodes along each I and J line to be equally spaced along a straight line

Geometric Ratios

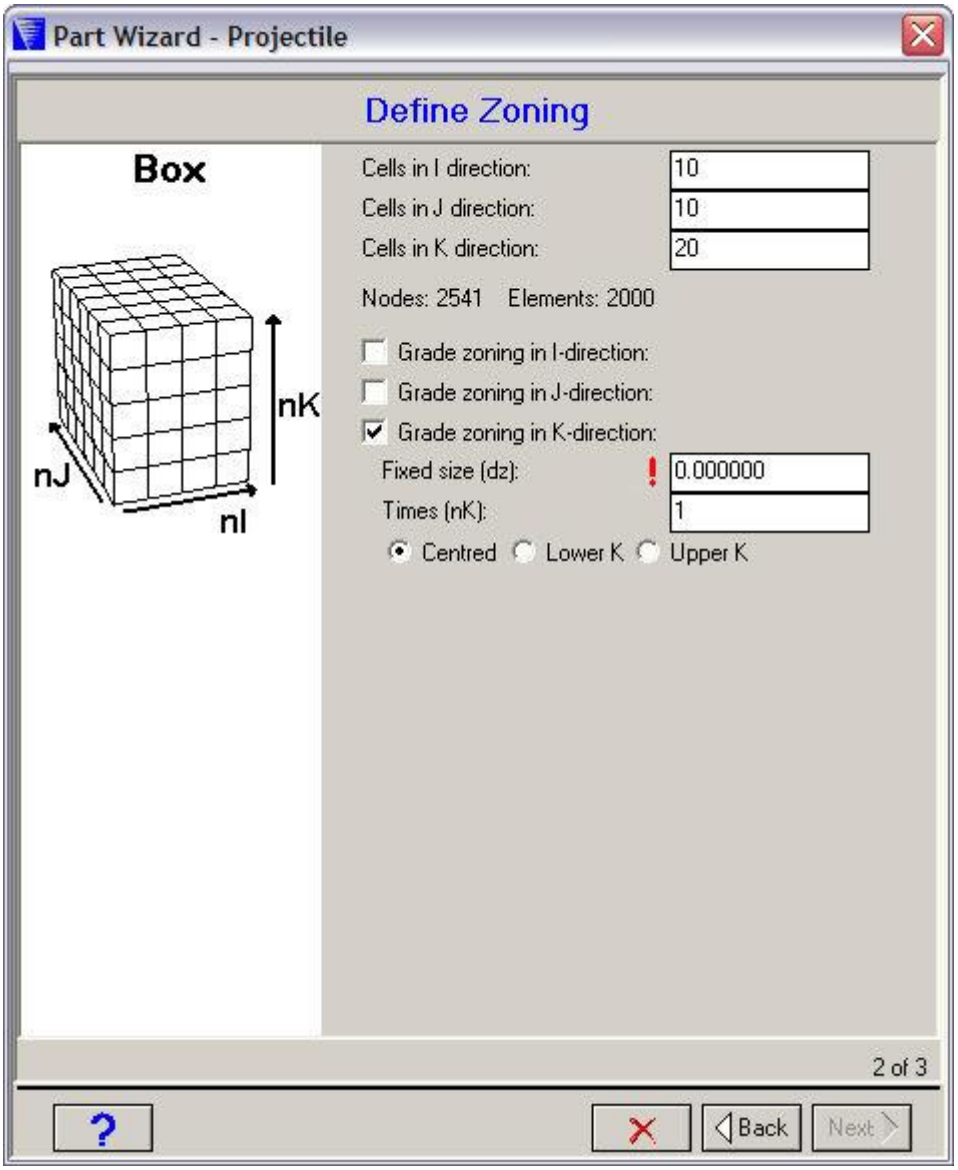
Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Define Zoning (3D)

This window lets you Define the zoning for your 3D Part.

Box



Cells

Enter the number of cells (elements) you want to use in the I, J and K directions in fields provided. The corresponding number of nodes and elements required for your part will be displayed below.

Grade zoning

Check the boxes if you want to grade your zoning in any direction.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

Fixed size

Enter the dimension you want to use for the fixed size cells in this field.

Times

Enter the number of fixed size cells you want to use in this field.

Position

Specify where you want to place the fixed size cells relative to the full I/J/K range.

Centered - at the center of the full I/J/K range
Lower - at the lower end of the I/J/K range
Upper - at the upper end of the I/J/K range

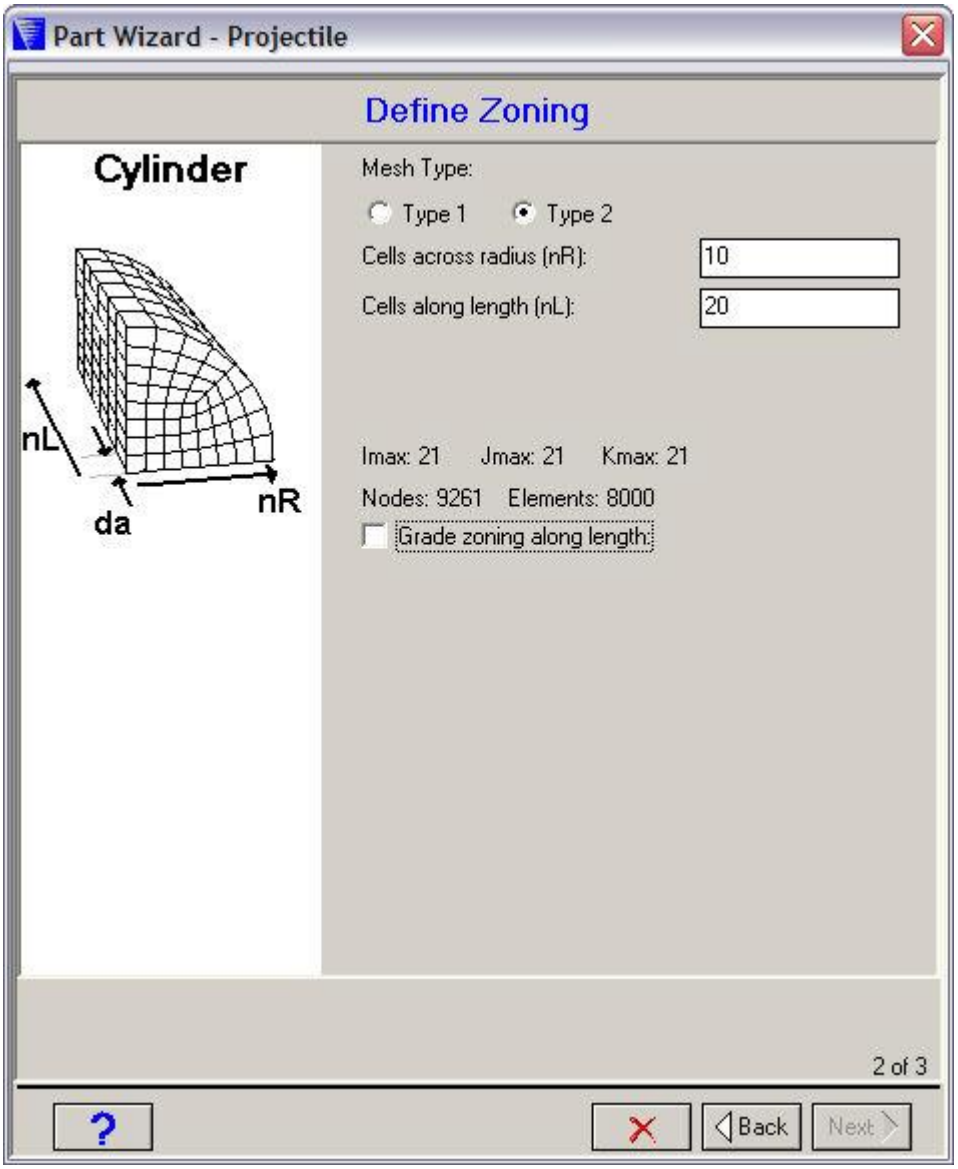
e.g. ten fixed size cells centered :

e.g. twelve fixed size cells at the lower end :

e.g. eight fixed size cells at the upper end :

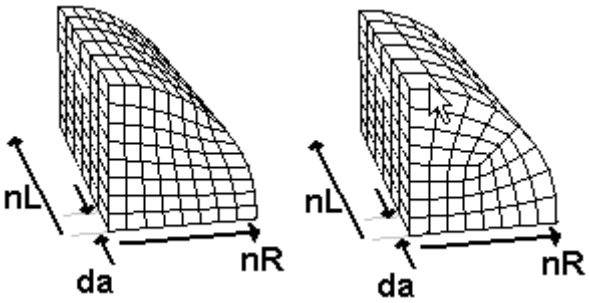
Cylinder

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Mesh Type

Select a Type 1 or Type 2 mesh for your cylinder.



Type 1 **Type 2**

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

All the elements of a type 2 mesh are reasonably rectangular, but about a quarter of the elements generated for this mesh are unused.

Cells across radius

Enter the number of cells (elements) you want to use across the radius of your cylinder in this field.

Cells along length

Enter the number of cells (elements) you want to use along the length of your cylinder in this field.

Number of elements

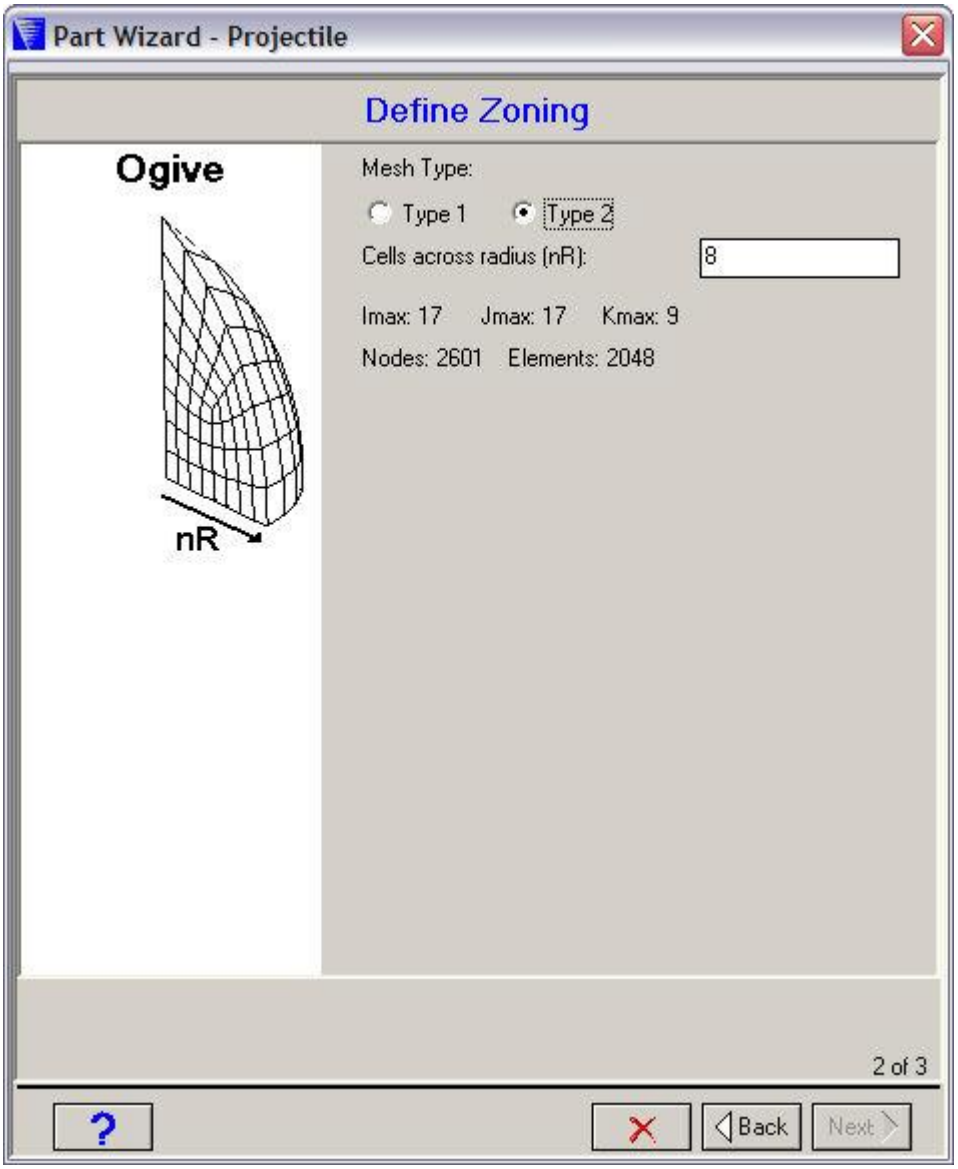
Displayed below the above input fields are the index space that will be used for your Part and the number of nodes and elements required.

Grade zoning along length

Check this box if you want to grade the zoning along the length of your cylinder (you will be asked to specify a **starting increment** to use).

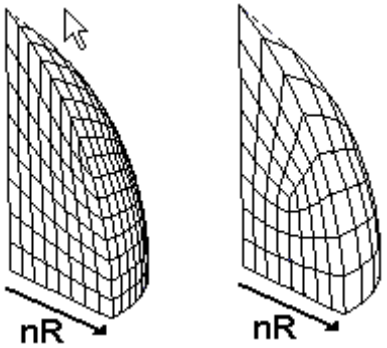
Ogive

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Mesh Type

Select a Type 1 or Type 2 mesh for your ogive.



Type 1 **Type 2**

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

All the elements of a type 2 mesh are reasonably rectangular, but some of the elements generated for this mesh are unused.

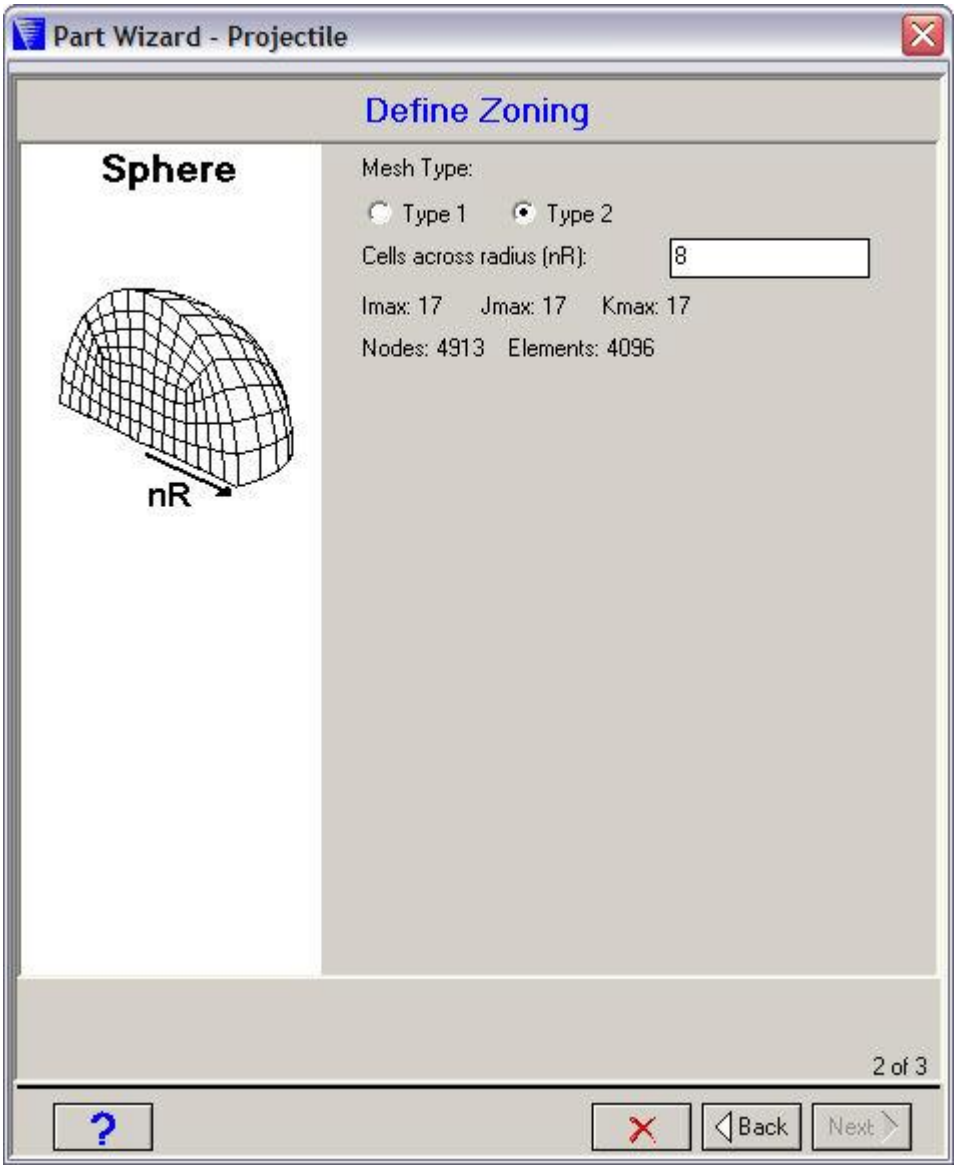
Cells across radius

Enter the number of cells (elements) you want to use across the radius of your ogive in this field.

Number of elements

Displayed below the input fields is the index space that will be used for your Part and the number of nodes and elements required.

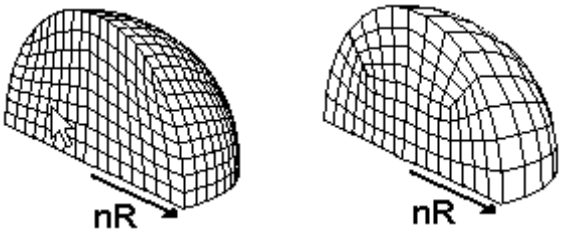
Sphere



Mesh Type

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select a Type 1 or Type 2 mesh for your sphere.



Type 1 **Type 2**

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but some of the elements generated for this mesh are unused.

Cells across radius

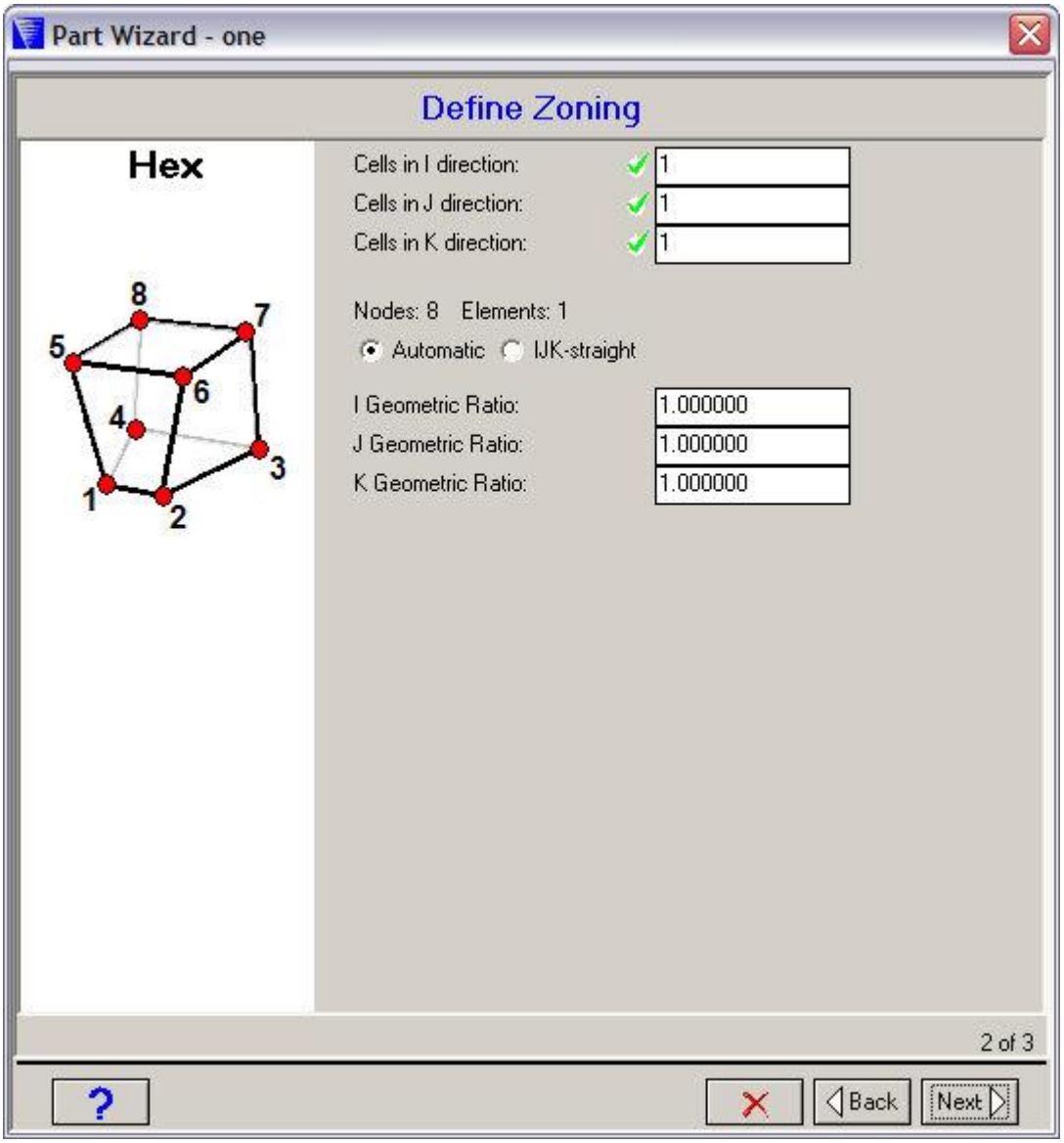
Enter the number of cells (elements) you want to use across the radius of your sphere in this field.

Number of elements

Displayed below the input fields is the index space that will be used for your Part and the number of nodes and elements required.

Hex

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Cells in I/J/K direction

Enter the number of cells (elements) you want to use in the I, J and K directions in fields provided. The corresponding number of nodes and elements required for your part will be displayed below.

Automatic

Select this option if you want the internal nodes to be spaced equipotentially.

IJK-Straight

Select this option if you want the nodes along each I, J and K line to be equally spaced along a straight line.

Geometric Ratios

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

Frag/Brick

Number of elements in each fragment/brick

Use these fields to set the number of elements per fragment/brick in the X, Y and Z directions.

Fill mortar with unused?

Check this box if you do not want the mortar to be represented as a material.

If you check this box, no elements will be generated for the mortar

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

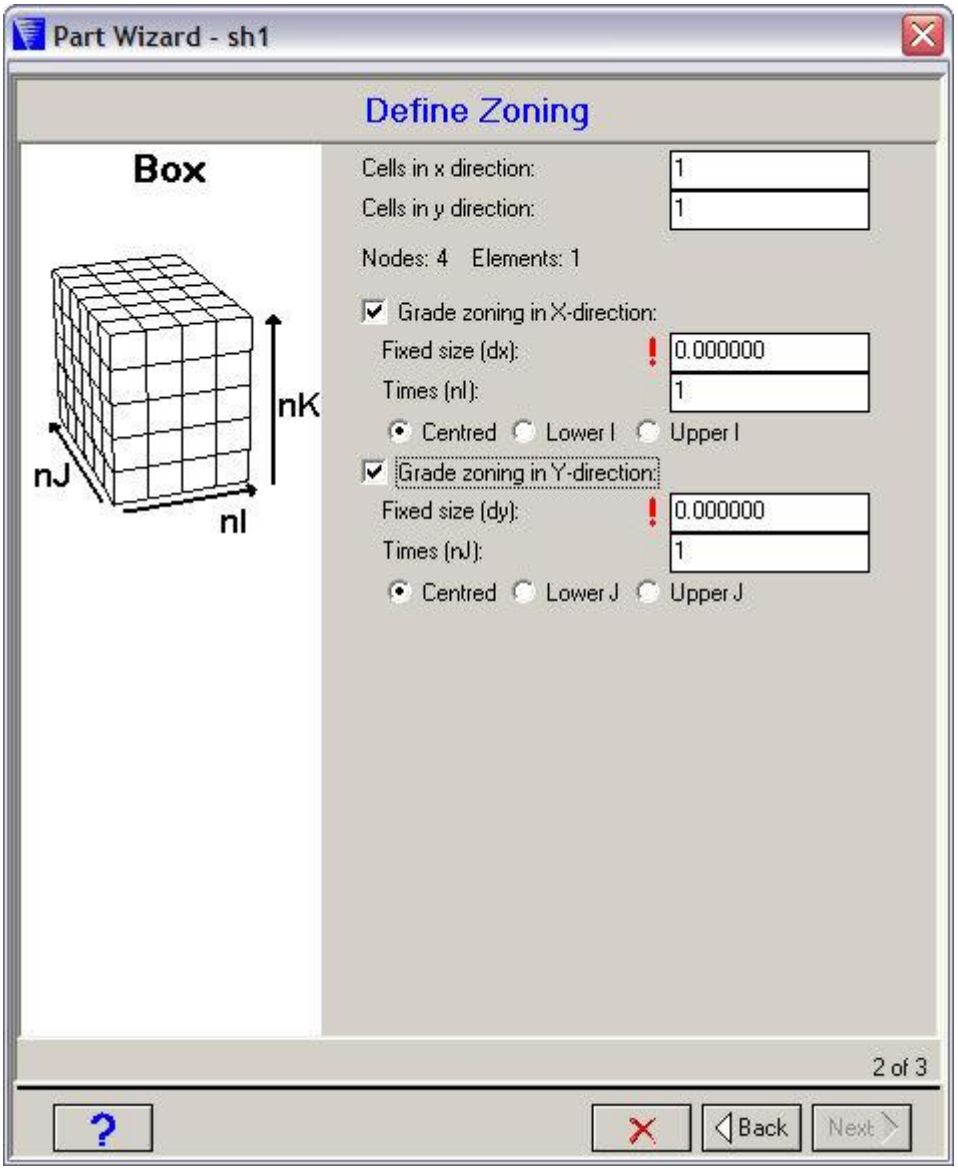
If you do not check this box, elements will be generated for the mortar (if the mortar thickness is small compared to the dimension of the brick elements, the time step is likely to be reduced significantly).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Define Zoning (3D Shell)

This window lets you Define the zoning for your Shell Part.

Plane



Cells

Enter the number of cells (elements) you want to use in the X, Y, Z directions in fields provided. The corresponding number of nodes and elements required for your part will be displayed below.

Grade zoning

Check the boxes if you want to grade your zoning in any direction.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the overall dimension of the box you specified in the previous window.

Fixed size

Enter the dimension you want to use for the fixed size cells in this field.

Times

Enter the number of fixed size cells you want to use in this field.

Position

Specify where you want to place the fixed size cells relative to the full I/J/K range.

Centered - at the center of the full I/J/K range
Lower - at the lower end of the I/J/K range
Upper - at the upper end of the I/J/K range

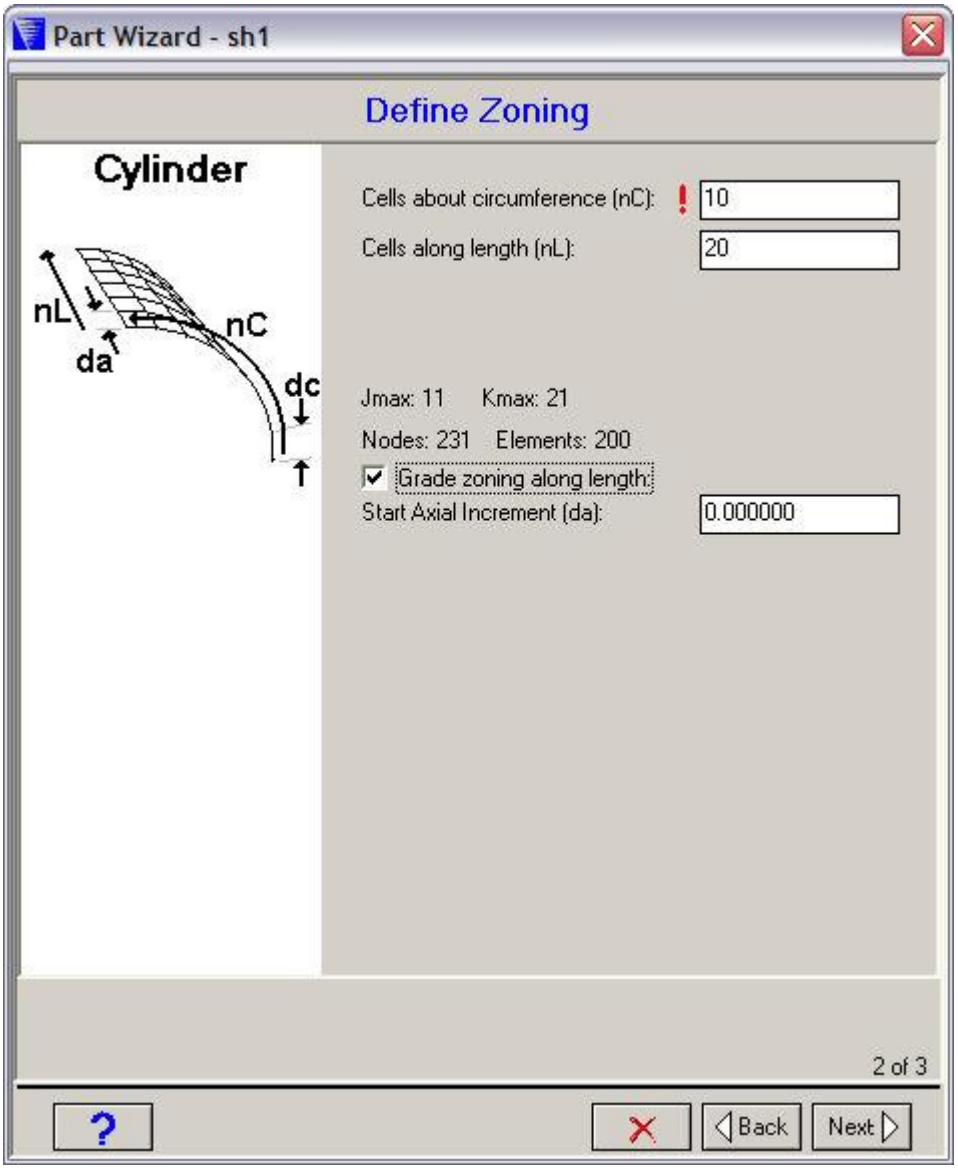
e.g. ten fixed size cells centered :

e.g. twelve fixed size cells at the lower end :

e.g. eight fixed size cells at the upper end :

Cylinder

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Cells about circumference

Enter the number of cells (elements) you want to use about the circumference of your cylinder in this field.

Cells along length

Enter the number of cells (elements) you want to use along the length of your cylinder in this field.

Number of elements

Displayed below the above two input fields is the index space that will be used for your Part and the number of nodes and elements required.

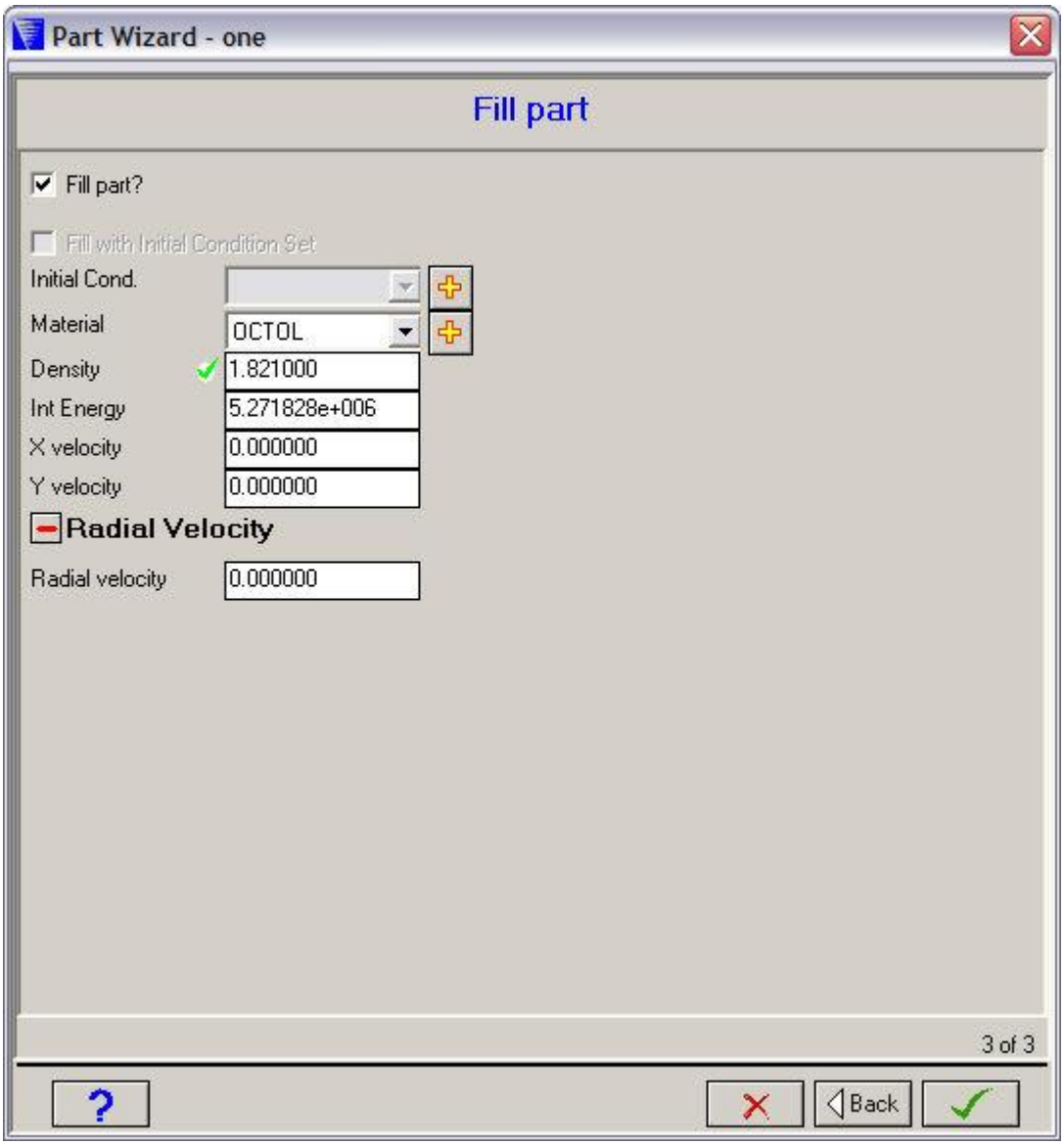
Grade zoning along length

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box if you want to grade the zoning along the length of your cylinder (you will be asked to specify a **starting increment** to use).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Fill Part (2D)



This window lets you fill the part you are creating.

Fill Part

Check this box if you want the Part Wizard to fill your Part.

The Part Wizard lets you fill your entire Part with a single material and initial values.

If you want to fill your part with multiple materials or initial conditions, this can be done by pressing the **Fill** button after completing the Part Wizard.

Fill with Initial Condition Set

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box if you want to fill your Part using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

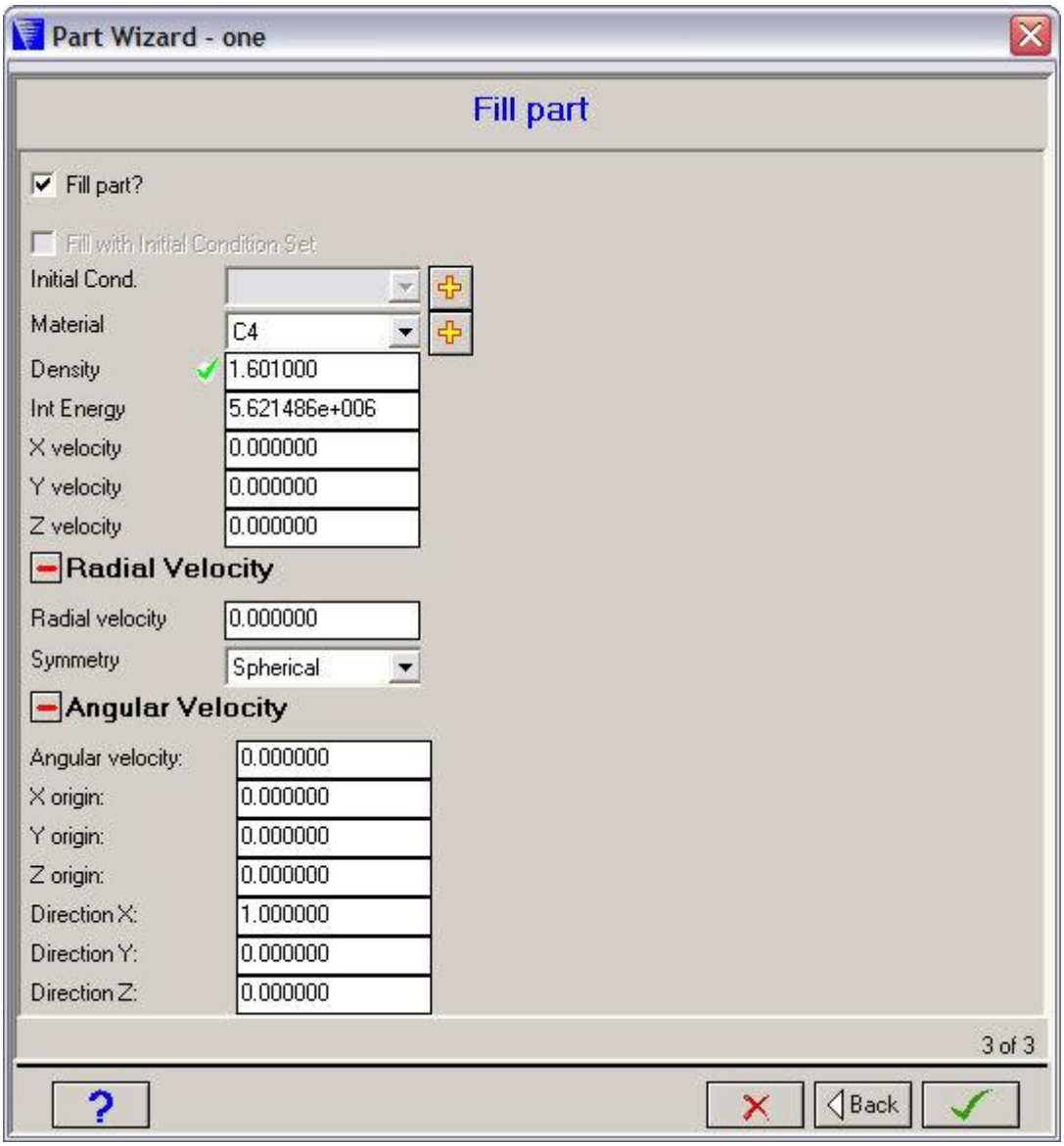
You can define new materials by pressing the **Materials** button on the Navigation Bar or by pressing next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (Density, Energy, Velocity).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Fill Part



This window lets you fill the part you are creating.

Fill Part

Check this box if you want the Part Wizard to fill your Part.

The Part Wizard lets you fill your entire Part with a single material and initial values.

If you want to fill your part with multiple materials or initial conditions, this can be done by pressing the **Fill** button after completing the Part Wizard.

Fill with Initial Condition Set

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box if you want to fill your Part using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

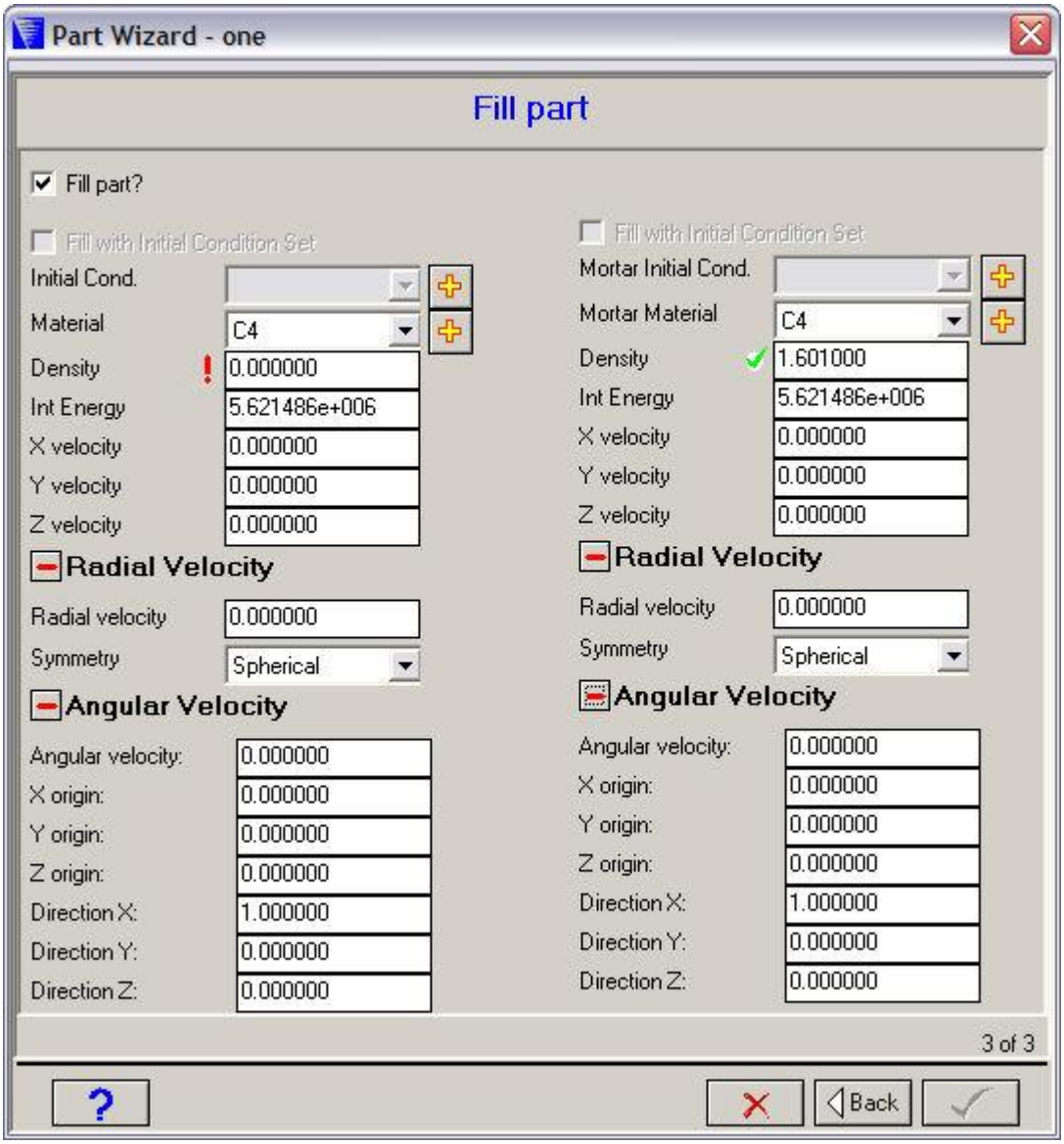
You can define new materials by pressing the **Materials** button on the Navigation Bar or by pressing next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (Density, Energy, Velocity).

Frag/Brick

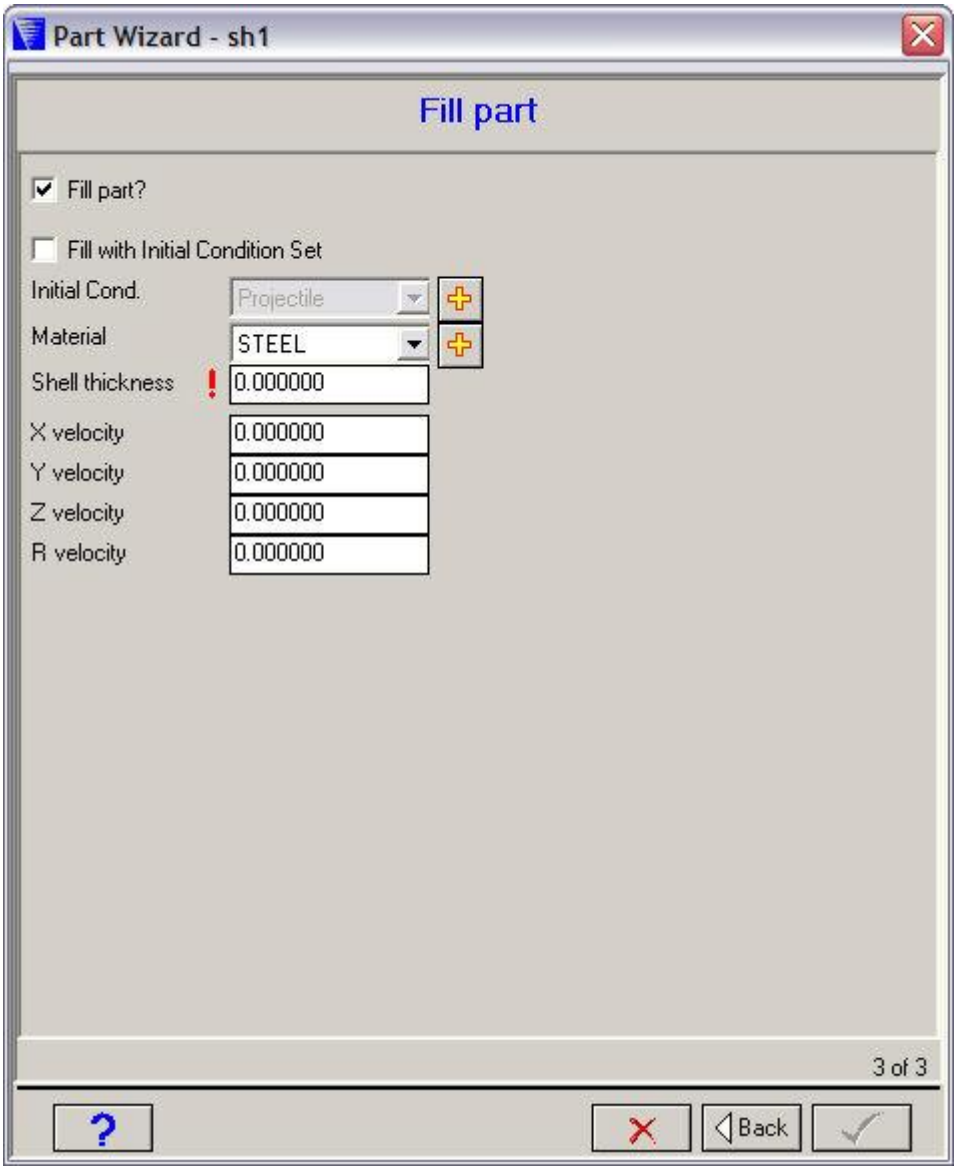
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Enter the material data for the brick and mortar materials

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - New - Fill Part (Shell)



This window lets you fill the shell part you are creating.

Fill Part

Check this box if you want the Part Wizard to fill your Part.

The Part Wizard lets you fill your entire Part with a single material and initial values.

If you want to fill your part with multiple materials or initial conditions, this can be done by pressing the **Fill** button after completing the Part Wizard.

Fill with Initial Condition Set

Check this box if you want to fill your Part using an Initial Condition Set.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

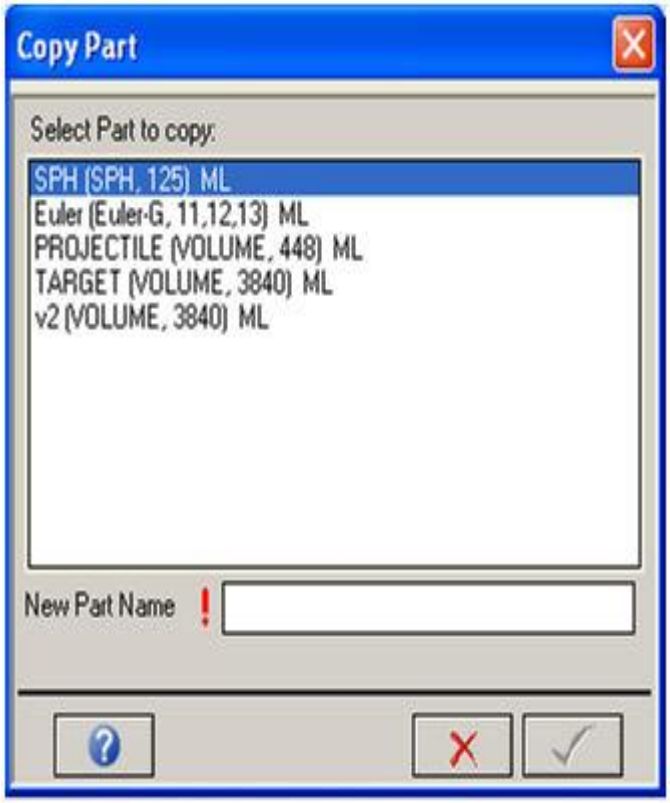
You can define new materials by pressing the **Materials** button on the Navigation Bar or by pressing next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (Shell Thickness, Velocity).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Copy



This window lets you copy an existing Part to a new Part.

Part List

This box lists all the existing Parts in your model. Select the part you want to copy from this list.

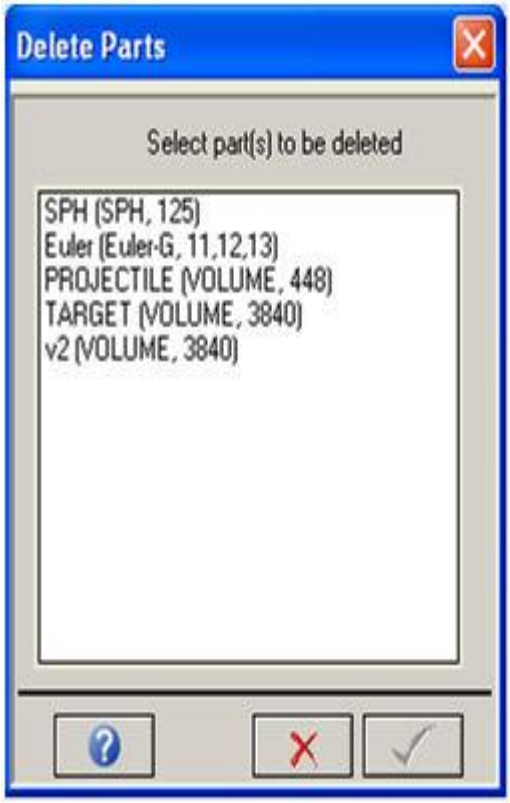
New Part Name

Enter the name of your new Part in this field.

Pressing creates your new part.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Delete

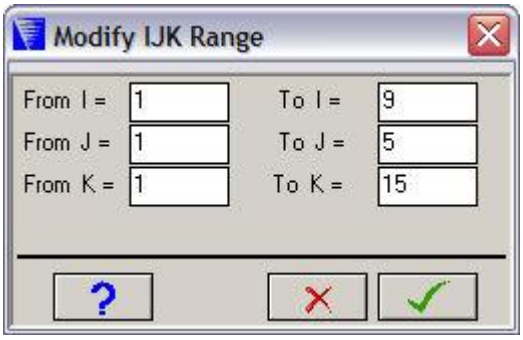


This window lets you delete Parts from your model.

Select the Part(s) you want to delete and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - IJK Range



This window lets you modify the IJK range of a structured Part.

Enter new IJK ranges for the Part in the fields provided.

If you cut a range, the corresponding elements are deleted from the Part.

If you extend a range, new (unused) elements are added to the Part.

Use zero or negative indices to add I, J or K lines to the lower end of ranges.

e.g. changing the I-range above to run From I = -4 To I = 12, will add 5 I-lines at the lower end of the Part and 3 I-lines at the upper end of the Part. The new I-range will then run from 1 to 17 (with the original I-lines moved to I=5 through I=14)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Activation

This window lets you set the activation and deactivation times for a structured Part. Activation / Deactivation times cannot be set for unstructured Parts. The default values ensure the Part is active for the entire calculation.

Activation Time

Enter the activation time for the Part in this field. The Part will not be computed until this activation time is reached.

Deactivation Time

Enter the deactivation time for the Part in this field. The Part will not be computed after this deactivation time is reached.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

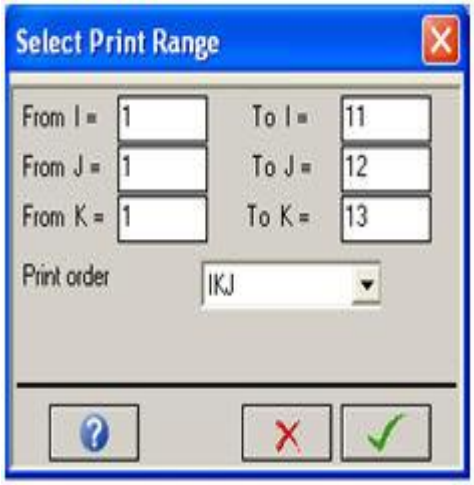
Parts - Rename

This window lets you rename a Part.

Type the new name for the Part in the field provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Print



This window lets you set the index range of a structured Part you want to use for grid data written to the print file (**.prt**) for this Part.

You can set the frequency at which this data is written (by cycle or times) in the **Output** panel (press **Output** on the Navigation Bar).

Index Range

Specify the index range for print data for this Part in the fields provided.

Print order

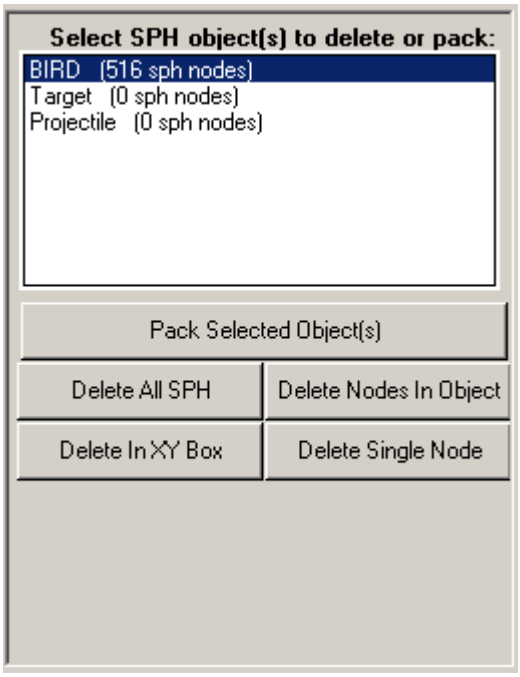
Use this pull-down menu to set the order in which data will be printed.

e.g. IKJ means J will vary faster than K and K will vary faster than I.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts-Pack(Fill)

When SPH parts are selected in the parts menu, pressing the "Pack (Fill)" button displays the following panel in the lower region of the parts menu.


Select SPH object(s) to delete or pack:
BIRD (516 sph nodes)
Target (0 sph nodes)
Projectile (0 sph nodes)
Pack Selected Object(s)
Delete All SPH
Delete Nodes In Object
Delete In XY Box
Delete Single Node

This panel allows you to fill SPH objects with SPH nodes.

Pack Selected Object(s)

Press this button to pack the selected objects with SPH nodes. You will be prompted for the fill material, the SPH node size and velocity to use for the packing operation.

Delete All SPH

Press this button to delete all SPH nodes in your model.

Delete Nodes in Object

Press this to delete the SPH nodes in the selected SPH objects.

Delete in XY Box

Press this to delete the SPH nodes in a box defined by the XYZ coordinates of the origin and the size of the box in the X, Y and Z directions.

Delete Single Node

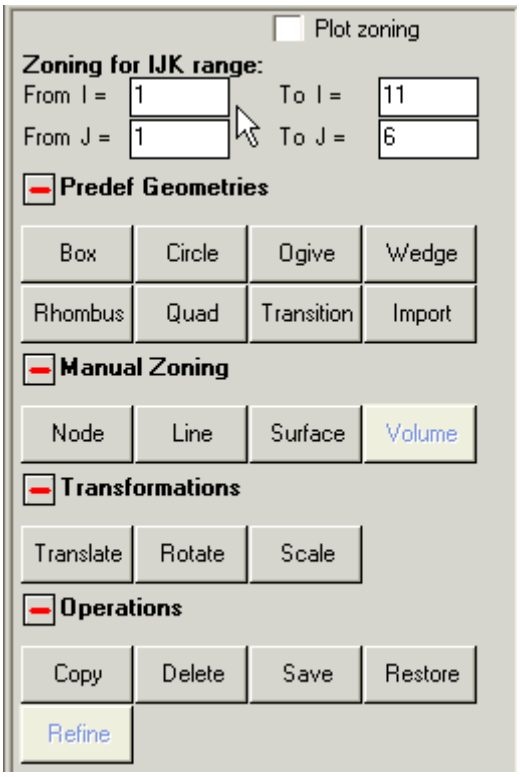
Press this to delete a single SPH node, by specifying its local IJK index. You can use the Examine option to interactively select SPH nodes, and determine their index.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

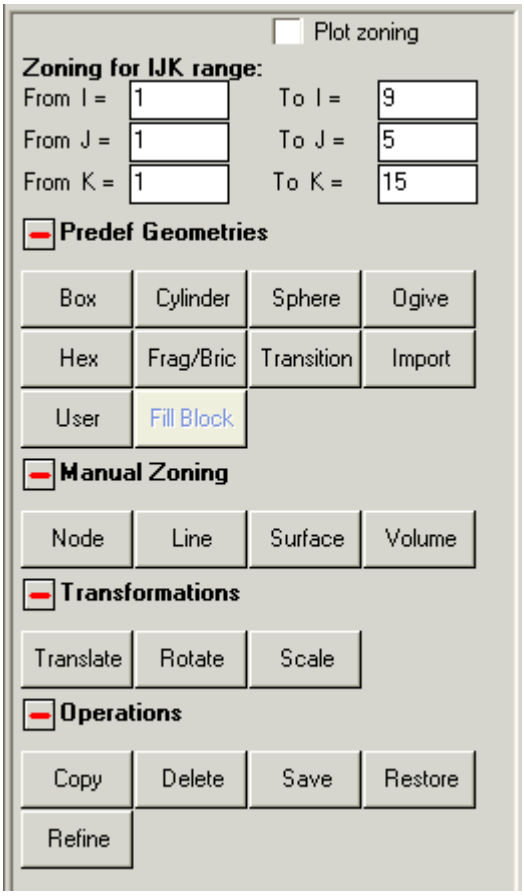
Zoning

Parts - Zoning

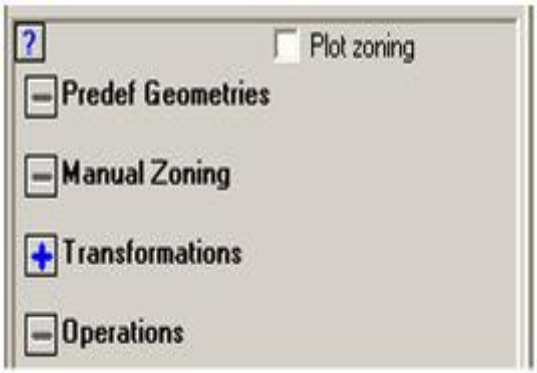
Pressing the **Zoning** button in the **Parts panel** displays the following dialog in the lower region of the panel :



2D-Structured



3D-Structured



2D & 3D-Unstructured

This dialog lets you define the zoning (grid coordinates) for the selected Part.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

For 2D & 3D unstructured Parts the geometry and meshing cannot be defined. The only option for unstructured Parts is the Transformation option, like translation, rotation or scaling.

Plot zoning

Standard plotting will only show elements that contain material, not unused elements.

Checking this box shows all elements, including unused elements.

Zoning for IJK range:

These fields let you specify the IJK range you want zoning operations to act on. The default values are for the entire Part.

Predef Geometries

Box
Press this button to zone a box.

Cylinder
Press this button to zone a cylinder.

Sphere
Press this button to zone a sphere.

Ogive
Press this button to zone an ogive.

Frag/Bric
Press this button to zone a fragment / brick wall.

Transition
Press this button to zone a grid transition

Import
Press this button to import zoning from an external file (**.zon**)

User
Press this button to generate zoning from a user subroutine.

Fill Block
Press this button to zone and fill an Euler-FCT grid using data from an external file.

Manual Zoning

These options let you build a grid by specifying nodes and generating lines, surfaces and 3D volumes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press + to reveal the following buttons :

Node
Press this button to define the coordinates of a node.

Line
Press this button to generate coordinates along a line in index space.

Surface
Press this button to generate the coordinates of an index plane.

Volume
Press this button to define the coordinates within the current IJK Range.

Transformations

These options let you translate, rotate and scale your zoning.
Press + to reveal the following buttons :

Translate
Press this button to translate zoning

Rotate
Press this button to rotate zoning

Scale
Press this button to scale zoning

Operations

These options let you perform various operations on your zoning.
Press + to reveal the following buttons :

Copy
Press this button to copy zoning from an existing Part to the current Part.

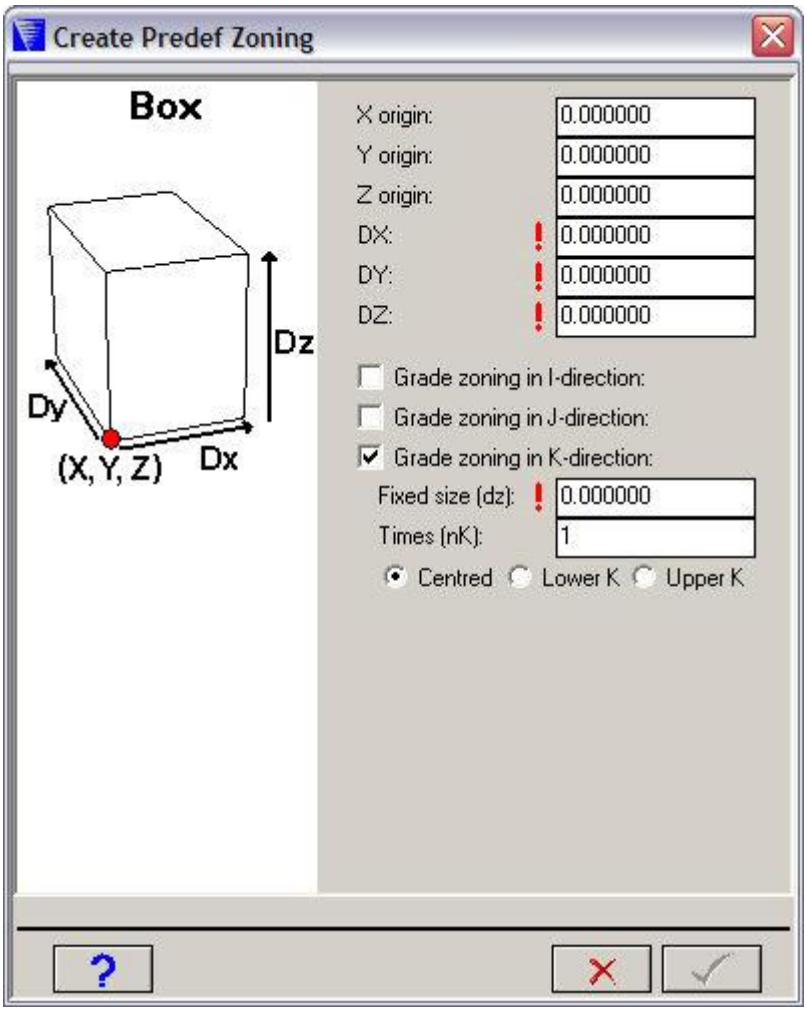
Delete
Press this button to delete zoning.

Save
Press this button to temporarily save your zoning.

Restore
Press this button to restore zoning you have temporarily saved.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Box



This window lets you generate a box with its axes parallel to the X, Y and Z axes.

Origin

Enter the X, Y and Z coordinates of the lower corner of the box in the fields provided.

Box Dimensions

Enter the dimensions of the box (DX, DY, DZ) in the fields provided

Grade zoning

Check the boxes if you want to grade your zoning in any direction.

If you choose this option you can specify a range of cells which have a fixed size. The remaining cells will be smoothly graded to fit the box dimensions.

Fixed size

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the dimension you want to use for the fixed size cells in this field.

Times

Enter the number of fixed size cells you want to use in this field.

Position

Specify where you want to place the fixed size cells relative to the full I/J/K range.

Centered - at the center of the full I/J/K range
Lower - at the lower end of the I/J/K range
Upper - at the upper end of the I/J/K range

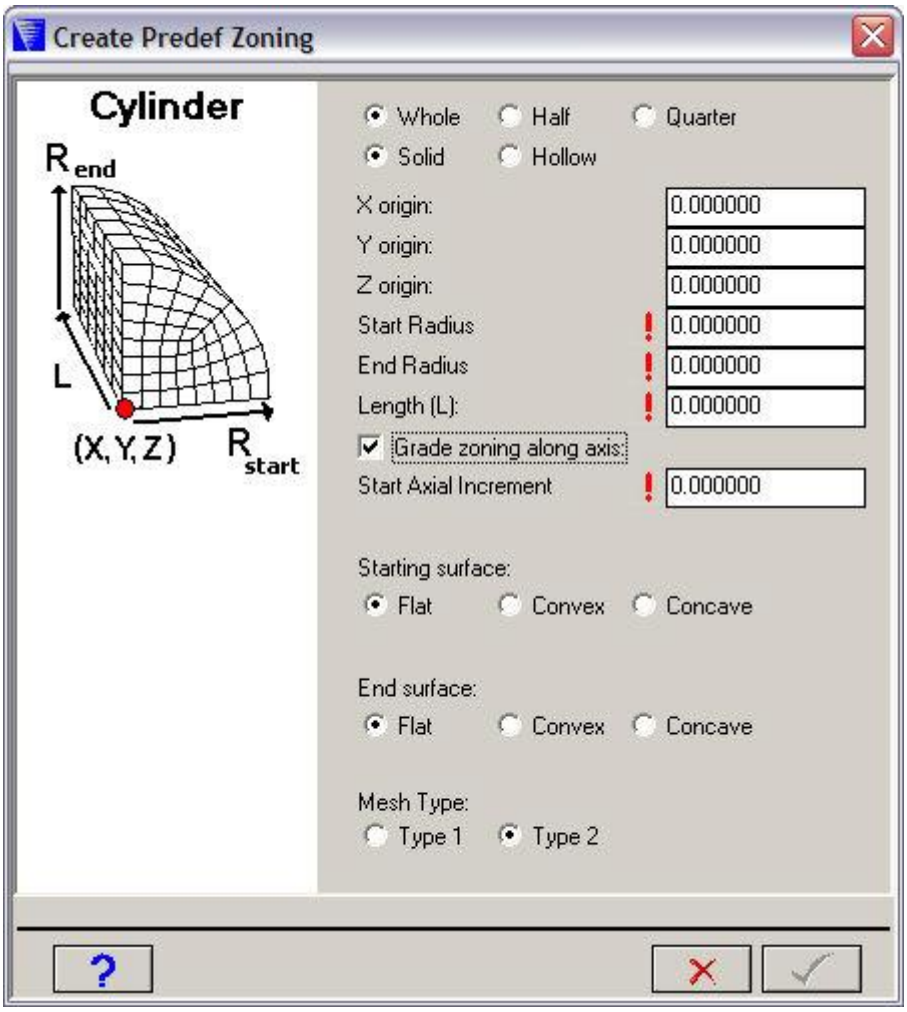
e.g. ten fixed size cells centered :

e.g. twelve fixed size cells at the lower end :

e.g. eight fixed size cells at the upper end :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Cylinder



This window lets you generate a cylinder with its axis parallel to the Z axis.

Section

Choose the section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow cylinder.

If you choose a hollow cylinder, you will have to specify inner and outer radii for the cylinder.

Origin

Enter the X, Y and Z coordinates for the center of the starting surface of your cylinder in the fields provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Start Radius / End Radius

You can specify different radii for the starting and end faces of your cylinder. For a regular cylinder (constant radius), set the starting radius equal to the end radius.

Length

Enter the length of your cylinder in this field.

Grade zoning along axis

Check this box if you want to grade the zoning along the length of your cylinder (you will asked to specify a geometric ratio to use).

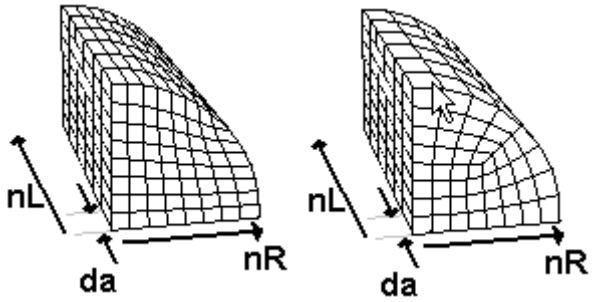
Start Surface / End surface

The starting and end surfaces of your cylinder can be flat, concave or convex.

If you select a concave or convex surface, you will have to specify its radius of curvature.

Mesh Type

Select a Type 1 or Type 2 mesh for your cylinder.



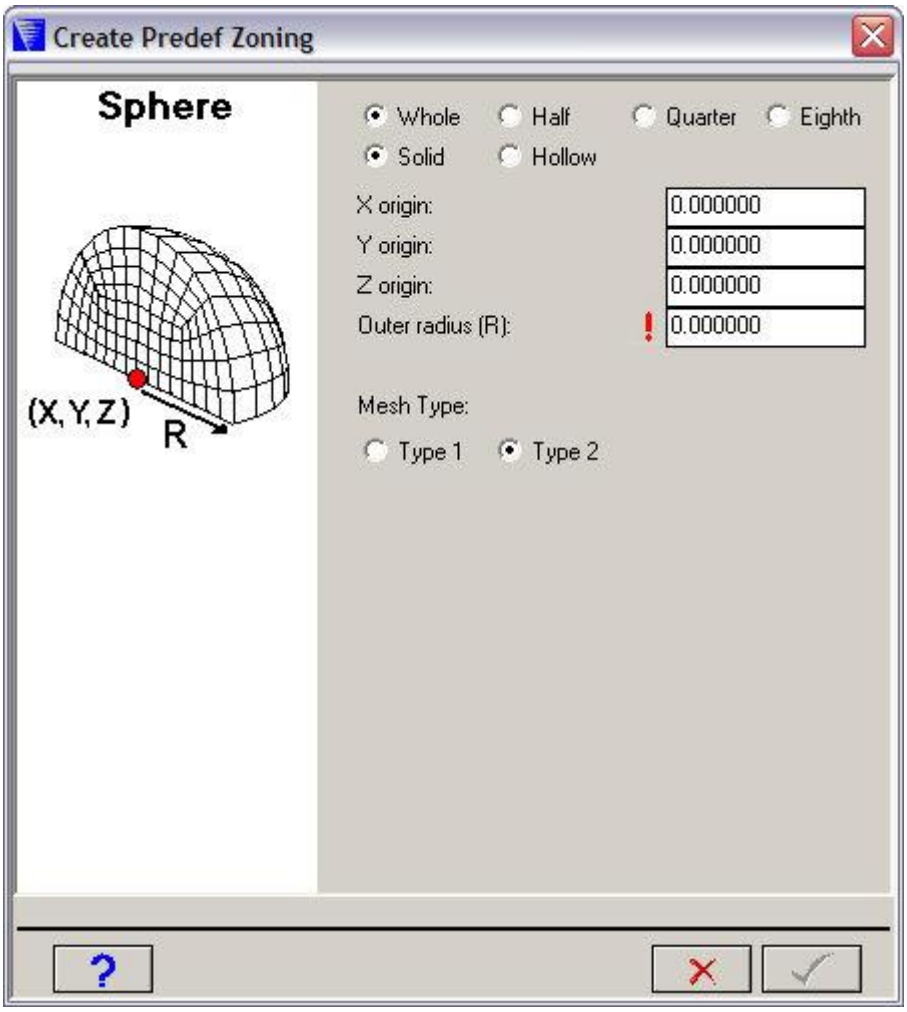
Type 1 **Type 2**

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but about a quarter of the elements generated for this mesh are unused.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Sphere



This window lets you generate a sphere.

Section

Choose the section you want to generate (Whole, Half, Quarter or Eighth). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow sphere.

If you choose a hollow sphere, you will have to specify inner and outer radii for the sphere.

Origin

Enter the X, Y and Z coordinates for the center of the sphere in the fields provided.

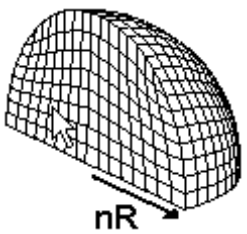
Radius

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

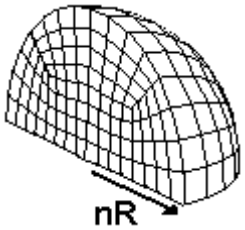
Enter the radius of the sphere in this field.

Mesh Type

Select a Type 1 or Type 2 mesh for your sphere.



Type 1



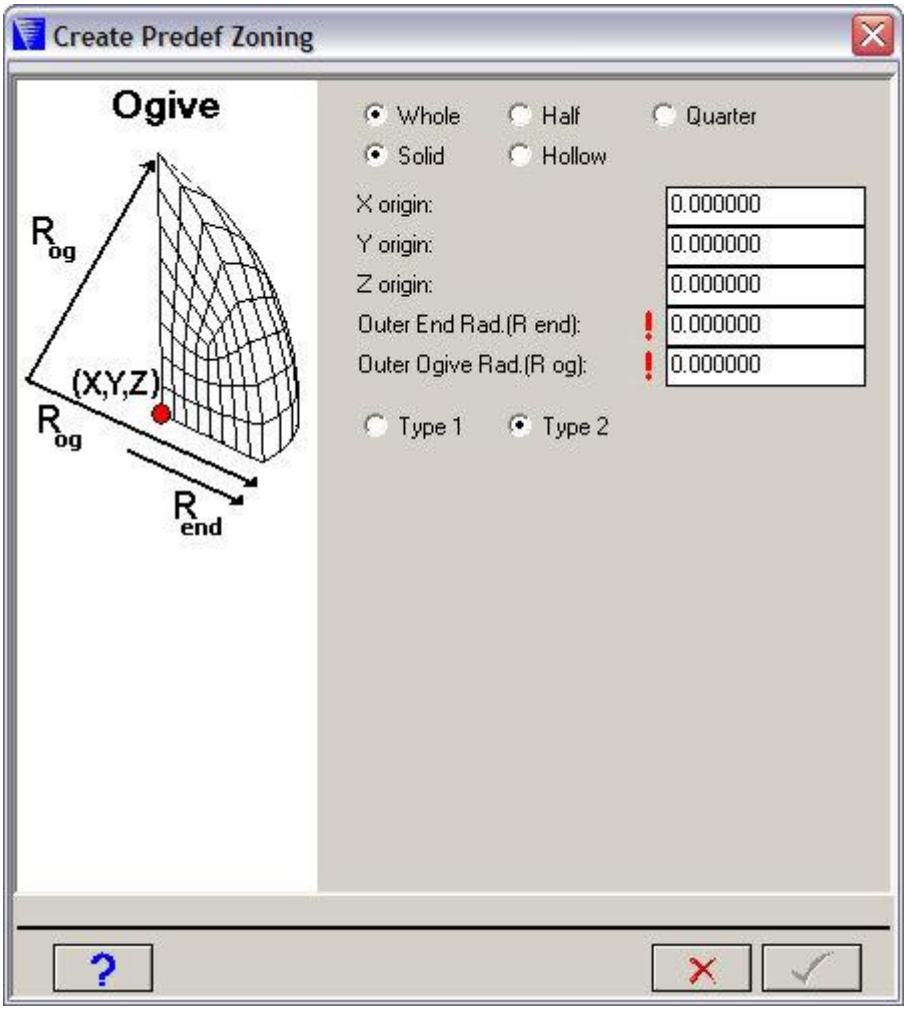
Type 2

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but some of the elements generated for this mesh are unused.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Ogive



This window lets you generate an ogive with its axis parallel to the Z axis.

Section

Choose the section you want to generate (Whole, Half or Quarter). This should match any symmetry planes you have defined for your model.

Solid or Hollow

Choose either a solid or hollow ogive.

If you choose a hollow ogive, you will have to specify inner and outer radii for the ogive.

Origin

Enter the X, Y and Z coordinates for the center of your ogive in the fields provided.

End Radius

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

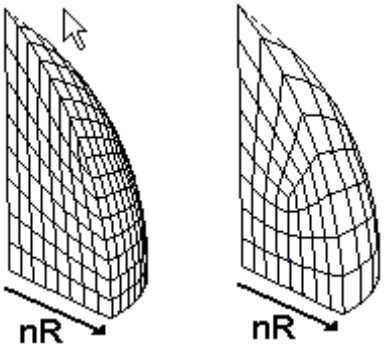
Enter the end radius of your ogive in this field.

Ogive Radius

Enter your ogive radius in this field.

Mesh Type

Select a Type 1 or Type 2 mesh for your ogive.



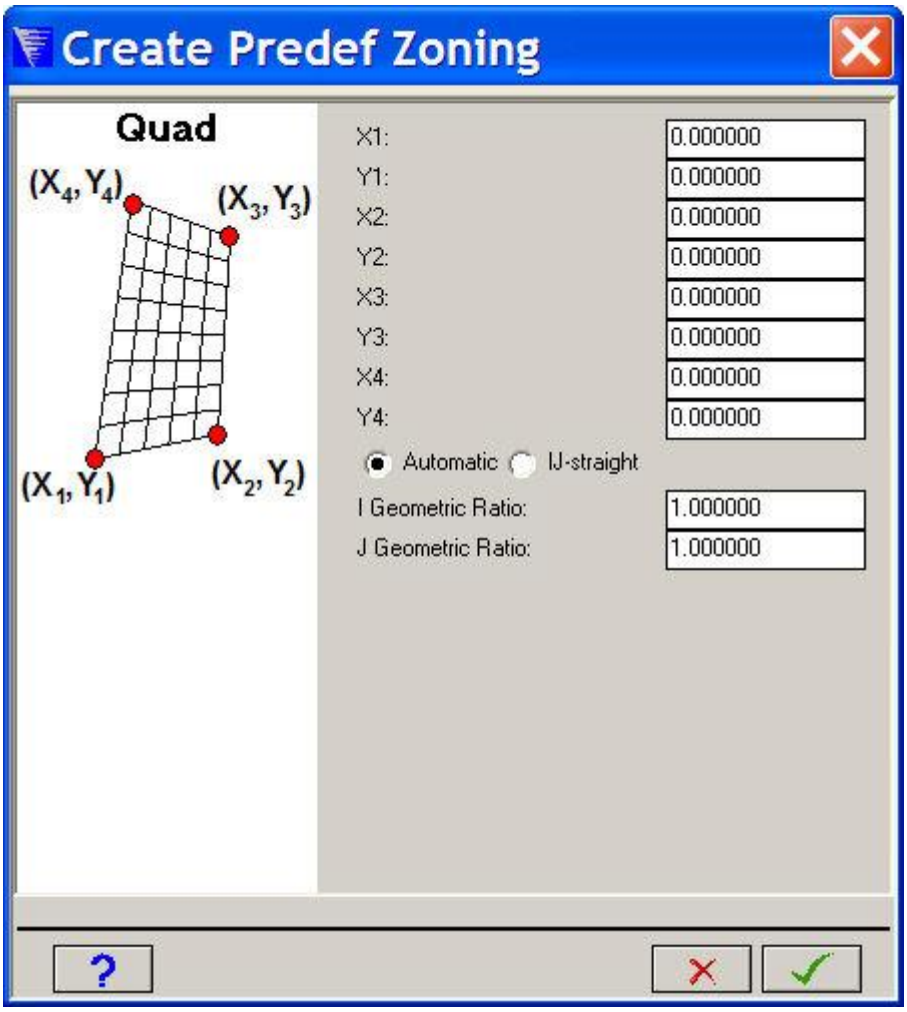
Type 1 **Type 2**

The type 1 mesh is more efficient in using elements, but some elements are not very rectangular (which can reduce the accuracy of your solution).

All the elements of a type 2 mesh are reasonably rectangular, but some of the elements generated for this mesh are unused.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Quad



Coordinates X1, Y1:

Enter coordinates of the first point of the quad.

Coordinates X2, Y2:

Enter coordinates of the second point of the quad.

Coordinates X3, Y3:

Enter coordinates of the third point of the quad.

Coordinates X4, Y4:

Enter coordinates of the fourth point of the quad.

Cells in I/J direction

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the number of cells (elements) you want to use in the I and J directions in fields provided. The corresponding number of nodes and elements required for your part will be displayed below.

Automatic

Select this option if you want the internal nodes to be spaced equipotentially.

IJ-Straight

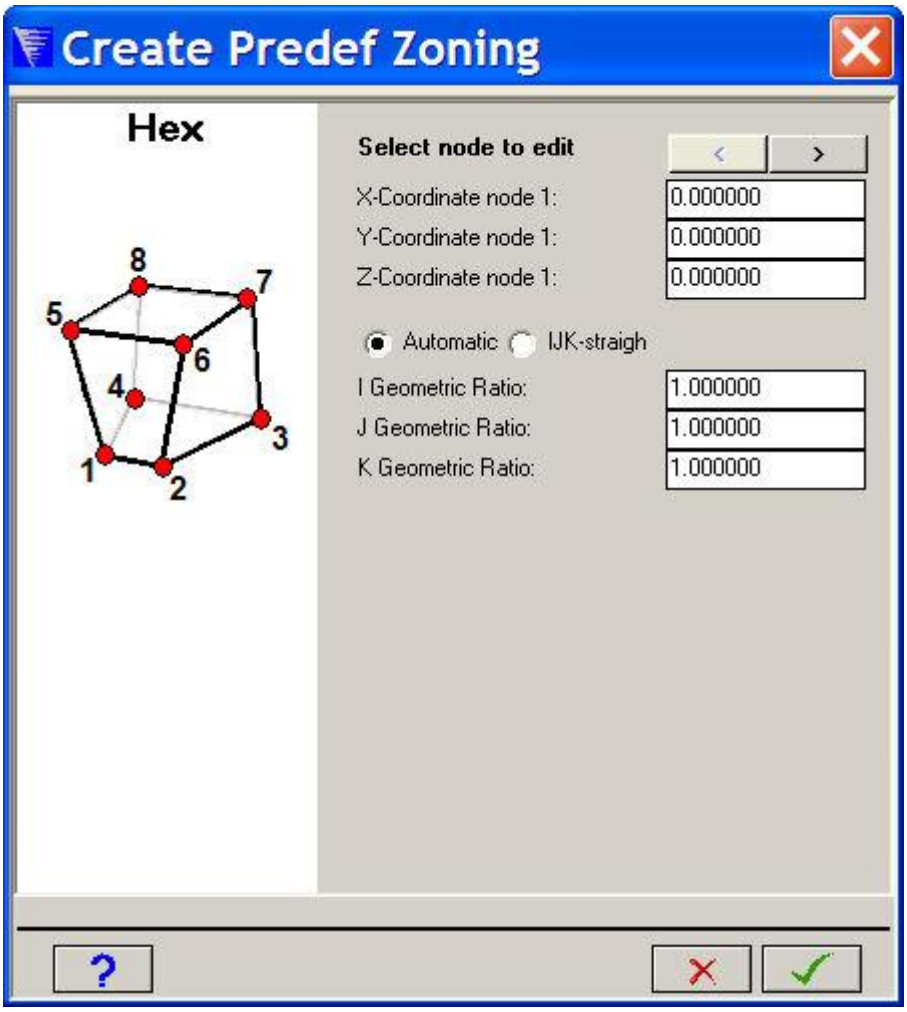
Select this option if you want the nodes along each I and J line to be equally spaced along a straight line

Geometric Ratios

Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Hex



Select Node to Edit

Use the arrow buttons to select each of the 8 nodes in turn.

Coordinates

Enter the X, Y and Z coordinates of the node.

Cells in I/J/K direction

Enter the number of cells (elements) you want to use in the I, J and K directions in fields provided. The corresponding number of nodes and elements required for your part will be displayed below.

Automatic

Select this option if you want the internal nodes to be spaced equipotentially.

IJK-Straight

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select this option if you want the nodes along each I, J and K line to be equally spaced along a straight line.

Geometric Ratios

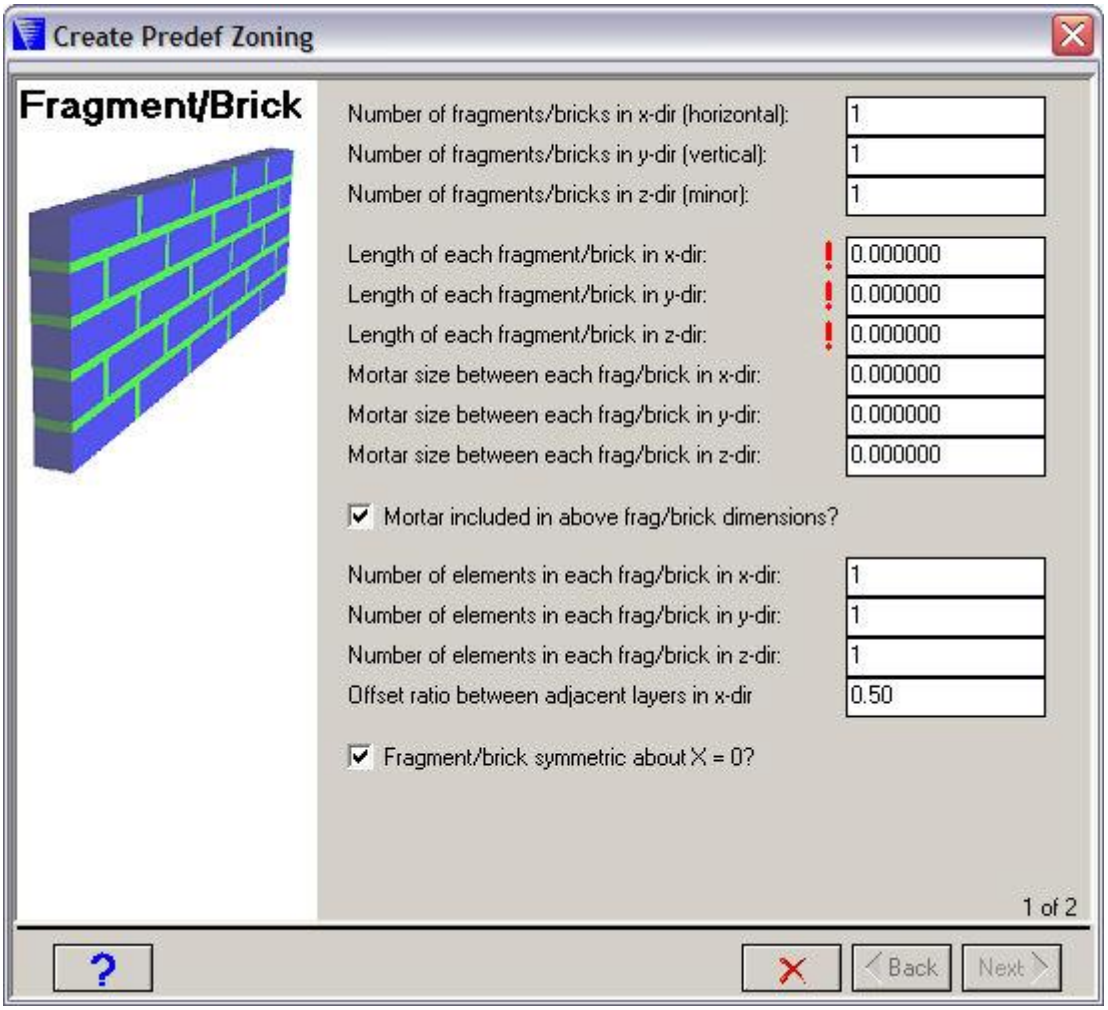
Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Frag/Bric (page 1)

Two dialog windows let you build a fragment or brick wall.

This first window lets you generate the zoning.


Create Predef Zoning
Fragment/Brick
Number of fragments/bricks in x-dir (horizontal):
Number of fragments/bricks in y-dir (vertical):
Number of fragments/bricks in z-dir (minor):
Length of each fragment/brick in x-dir:
Length of each fragment/brick in y-dir:
Length of each fragment/brick in z-dir:
Mortar size between each frag/brick in x-dir:
Mortar size between each frag/brick in y-dir:
Mortar size between each frag/brick in z-dir:
Mortar included in above frag/brick dimensions?
Number of elements in each frag/brick in x-dir:
Number of elements in each frag/brick in y-dir:
Number of elements in each frag/brick in z-dir:
Offset ratio between adjacent layers in x-dir
Fragment/brick symmetric about X = 0?
1 of 2
Back
Next

The fragment/brick wall is built with its axes parallel to the X, Y and Z axes.

The X direction runs the length of the wall.

The Y direction runs the height of the wall.

The Z direction runs through the thickness of the wall.

Number of fragments/bricks

Use these fields to set the number of fragments/bricks in the X, Y and Z directions.

Length of each fragment/brick

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Use these fields to set the length of a single brick/fragment in the X, Y and Z directions.

Mortar size between each fragment/brick

Use these fields to set the mortar size (or gap) between fragments/bricks.

Mortar included in dimension?

Check this box if you want the mortar size to be included in the length of each brick/fragment.

Number of elements in each fragment/brick

Use these fields to set the number of elements per fragment/brick in the X, Y and Z directions.

Offset ratio

Set the offset ratio for the fragments/bricks in this field.
This is how the bricks are staggered along the length of the wall.
e.g. if the offset is 0.5 (the default), the bricks will be staggered halfway across each other as shown in the above figure.

Fragment/brick symmetric about X=0?

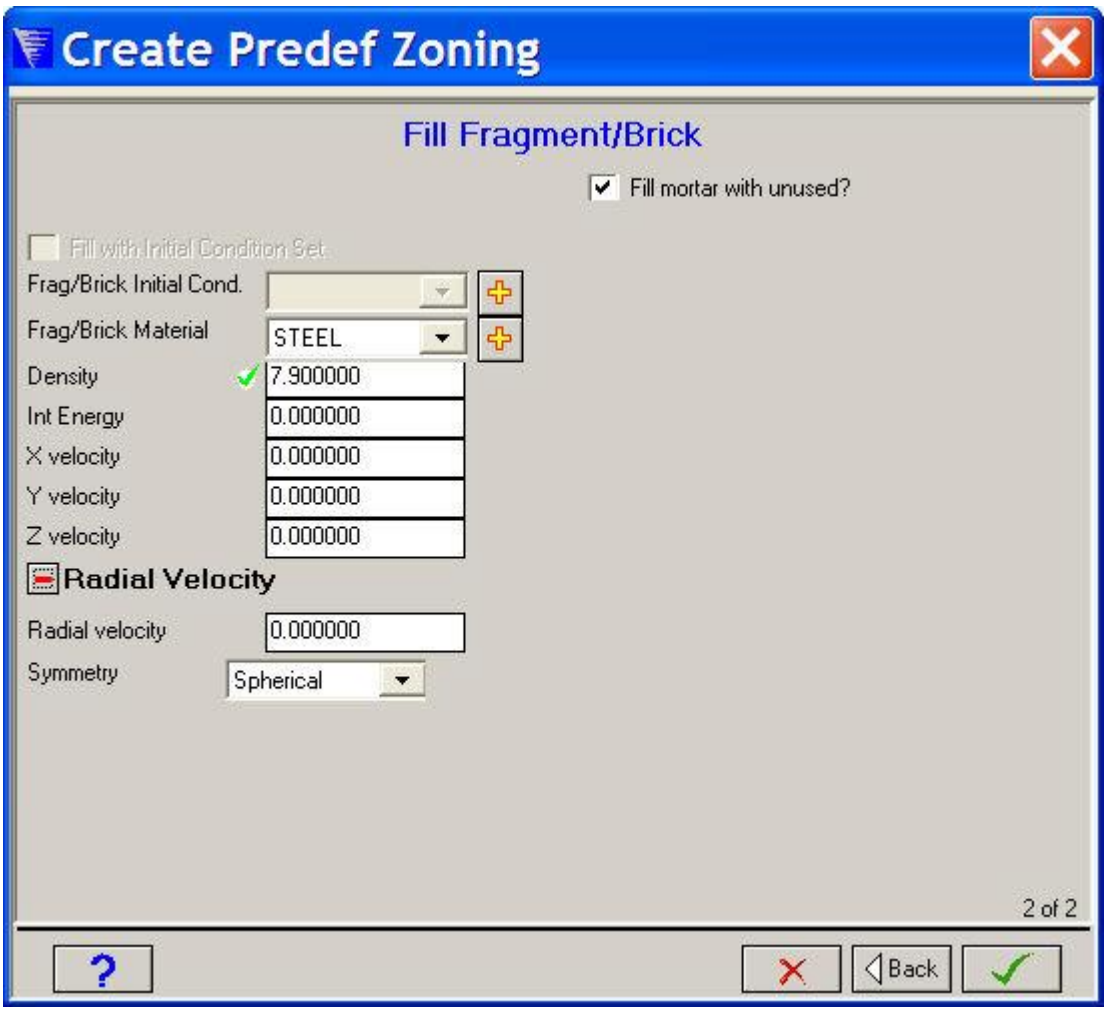
Check this box if you want the wall to be symmetric about X=0.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Frag/Bric (page 2)

Two dialog windows let you build a fragment or brick wall.

This second window lets you fill the wall grid with materials and initial conditions.



Fill mortar with unused?

Check this box if you do not want the mortar to be represented as a material.

If you check this box, no elements will be generated for the mortar.

If you do not check this box, elements will be generated for the mortar (If the mortar thickness is small compared to the dimension of the brick elements, the time step is likely to be reduced significantly).

Fill with Initial Condition Set

Check this box if you want to fill your wall Part using an Initial Condition Set.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

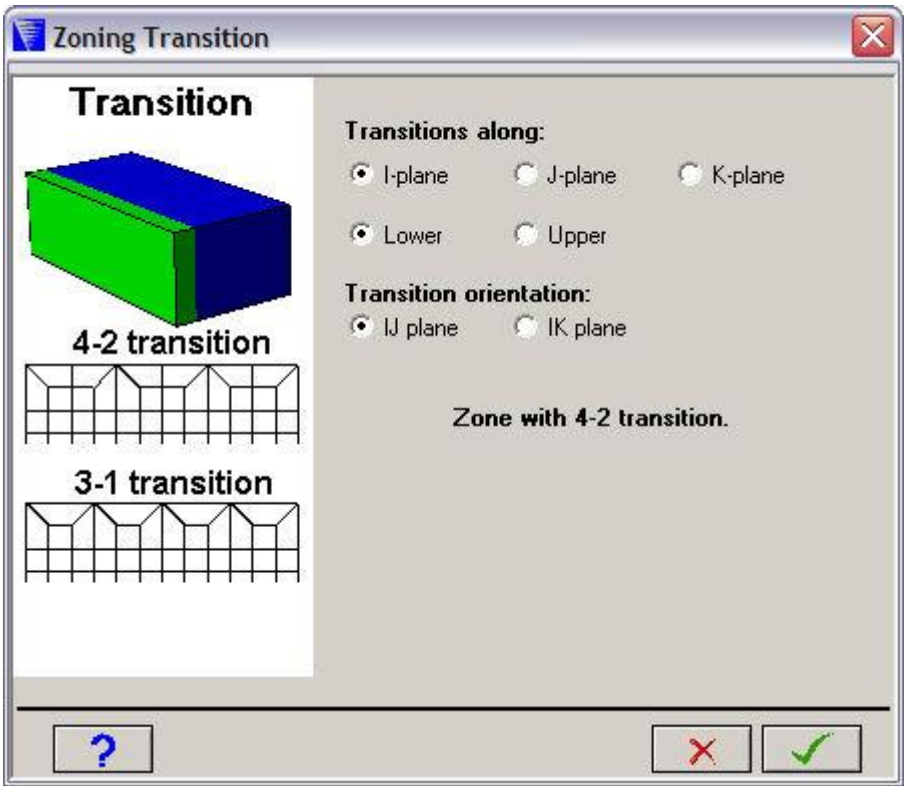
You can define new materials by pressing the **Materials** button on the Navigation Bar or by pressing next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (Density, Energy, Velocity).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Transition



This window lets you create a transition at any surface of the block you are zoning.

Transitions along:

Specify at which of the six surfaces of the block you want your transition to be created, by selecting and I, J or K-plane and whether it is the corresponding lower or upper surface of the block.

Transition orientation:

Select the orientation of the transition.

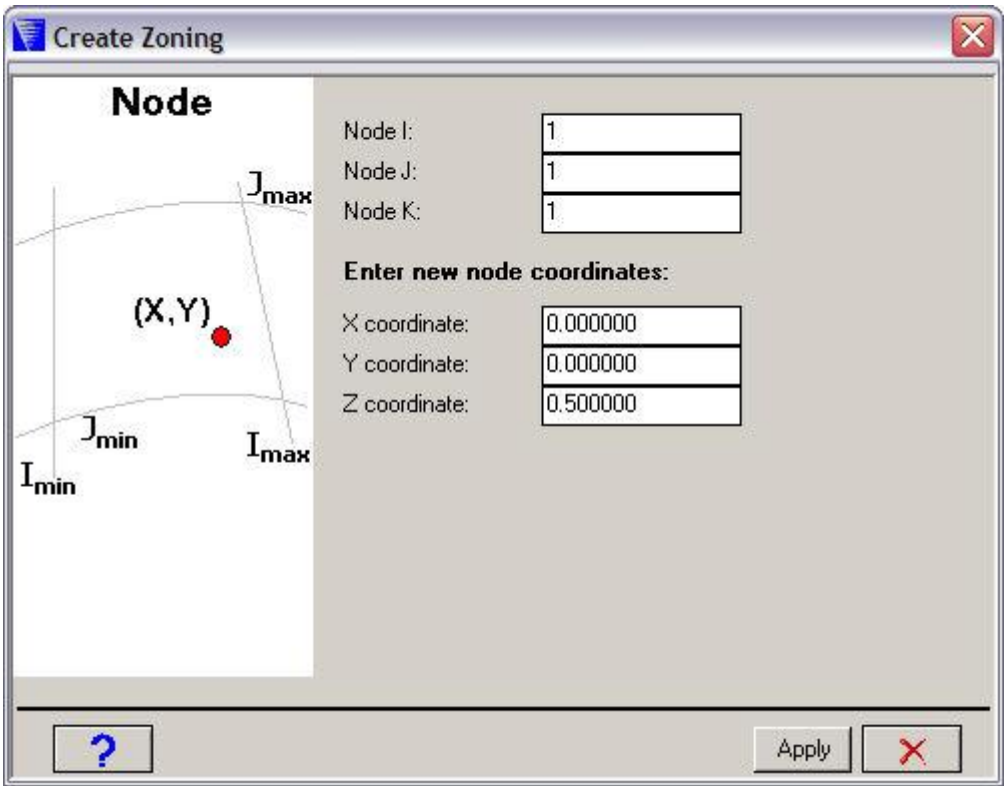
Transition Types

The image above shows you the two transition types you can use, namely a 4-2 transition and a 3-1 transition. To use a particular type, the index range in the direction of the transition orientation must be a multiple of 4 or 3 respectively.

A message will tell if a valid transition type exists for the boundary you have defined. If more than one type is valid, you will be asked to choose the transition type you want to use.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Node



This window lets you specify the coordinates of a single node.

Node I/J/K:

Enter the I, J and K index to select the node you want to specify coordinates for in this field.

Old Coordinates

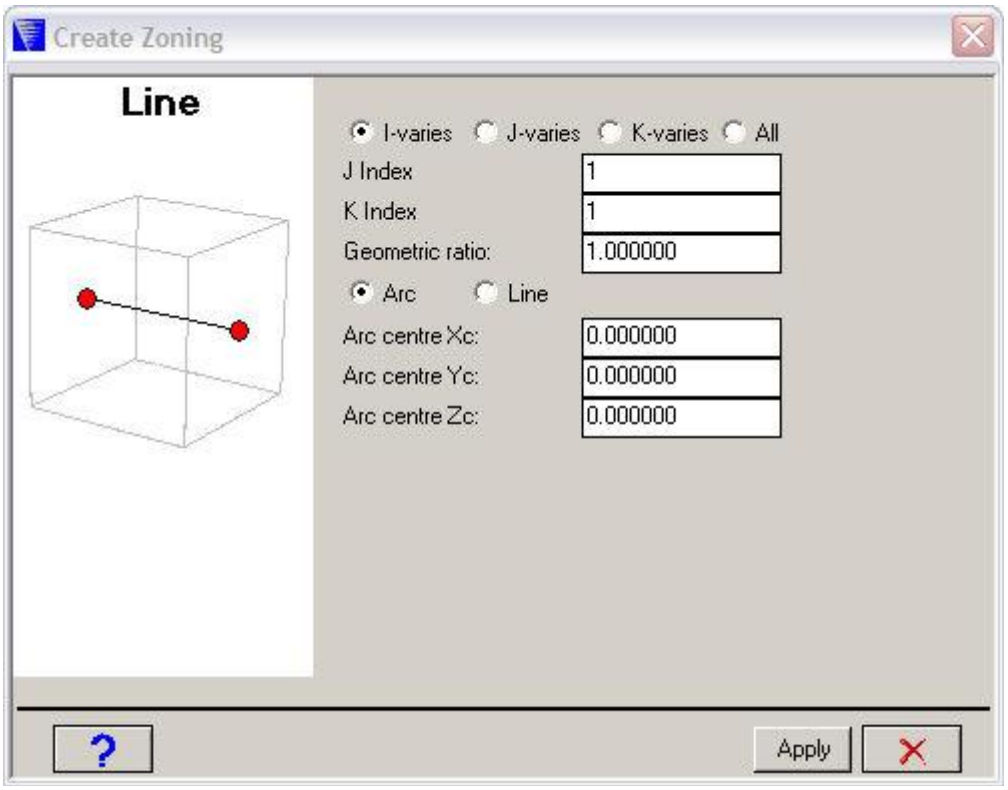
The current (old) coordinates of the selected node are displayed for your information.

New Coordinates

Enter the new coordinates for the selected node in these fields.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Line



This window lets you generate nodes along a line in index space from its two end points. If either of its two end nodes has not been defined, you are asked to input the coordinates of the node.

I/J/K-varies

Select the index direction of the line you want to generate. The line will be generated across the IJK range in the selected direction.

All

Select this option if you want to generate nodes for all 12 edges of the IJK range you have specified.

Geometric Ratios

Input geometric ratios in the fields provided if you want to grade the zoning instead of generating equally spaced nodes.

Line

Select this option if you want to generate the nodes along a straight line.

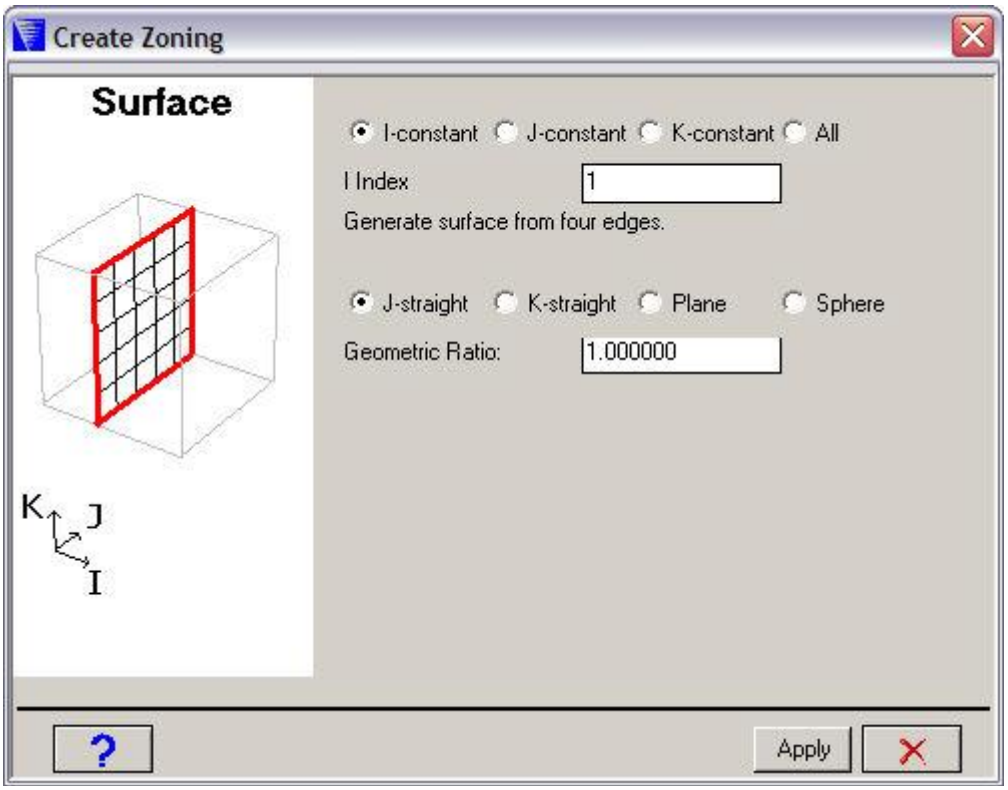
Arc

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select this option if you want to generate the nodes along the arc of a circle (you will be asked to specify the center of the circle).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Surface



This window lets you zone a surface of the IJK region you selected (i.e. an index plane with I, J or K constant) from nodes already defined along its four edges.

I/J/K-constant

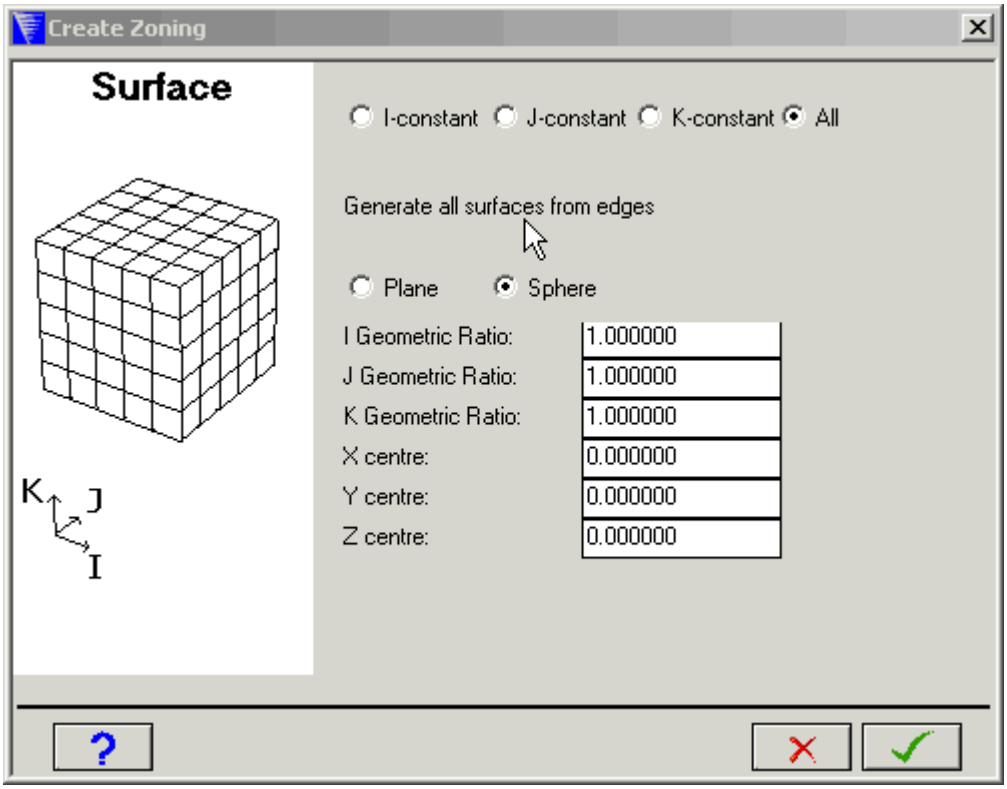
Select whether you want to zone a surface with a constant I, J or K index.

All

Select this option if you want to zone **all six outer surfaces** of the IJK range you have selected.

The dialog window will then look like this:

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Index (not for **All**)

Select the I, J or K index for the surface.

I/J/K-straight (not for **All**)

Select one of these options if you want surface nodes to be generated along straight I, J or K lines.

Plane

Select this option if you want surface nodes to be generated equipotentially in a plane.

Sphere

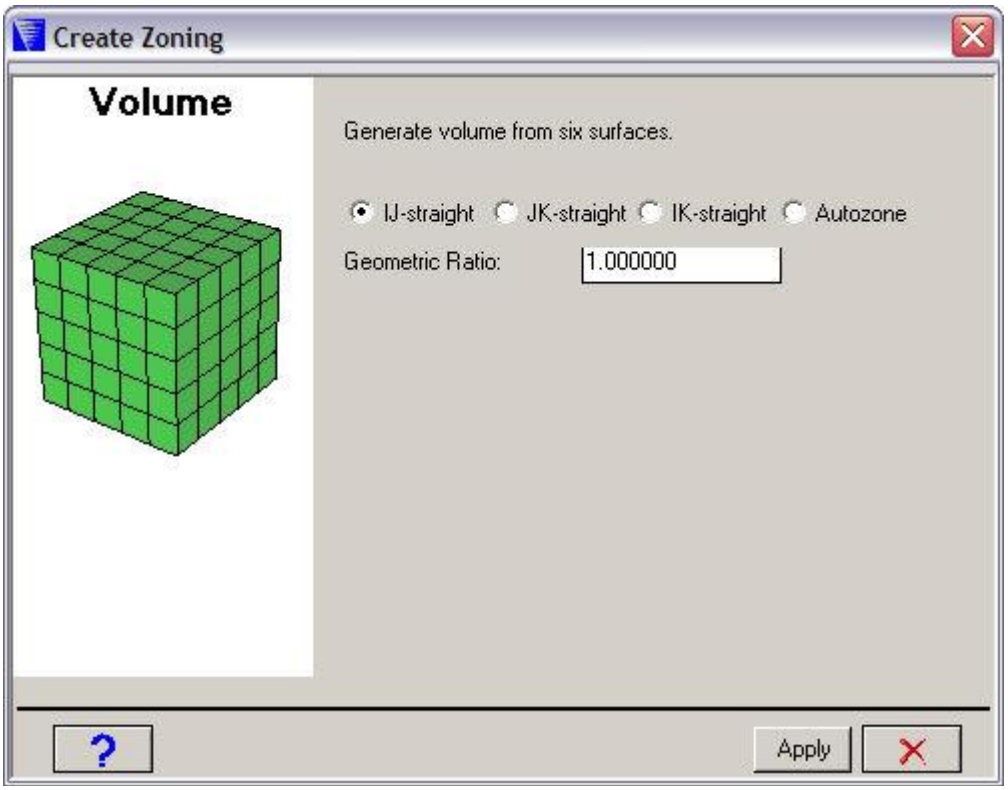
Select this option if you want surface nodes to be generated equipotentially on the surface of a sphere (you will be asked to specify the center of the sphere).

Geometric Ratios

Set geometric ratios in the fields provided if you want to grade the zoning instead of equally spacing the nodes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Volume



This window lets you zone the interior volume of the IJK region you selected from nodes already defined on its six faces.

IJ-straight

Select this option if you want to generate the interior volume so that nodes with the same (I,J) indices lie equally spaced along a straight line

JK-straight

Select this option if you want to generate the interior volume so that nodes with the same (J,K) indices lie equally spaced along a straight line

IK-straight

Select this option if you want to generate the interior volume so that nodes with the same (I,K) indices lie equally spaced along a straight line

Autozone

Select this option if you want to generate the interior volume so that all nodes are spaced equipotentially.

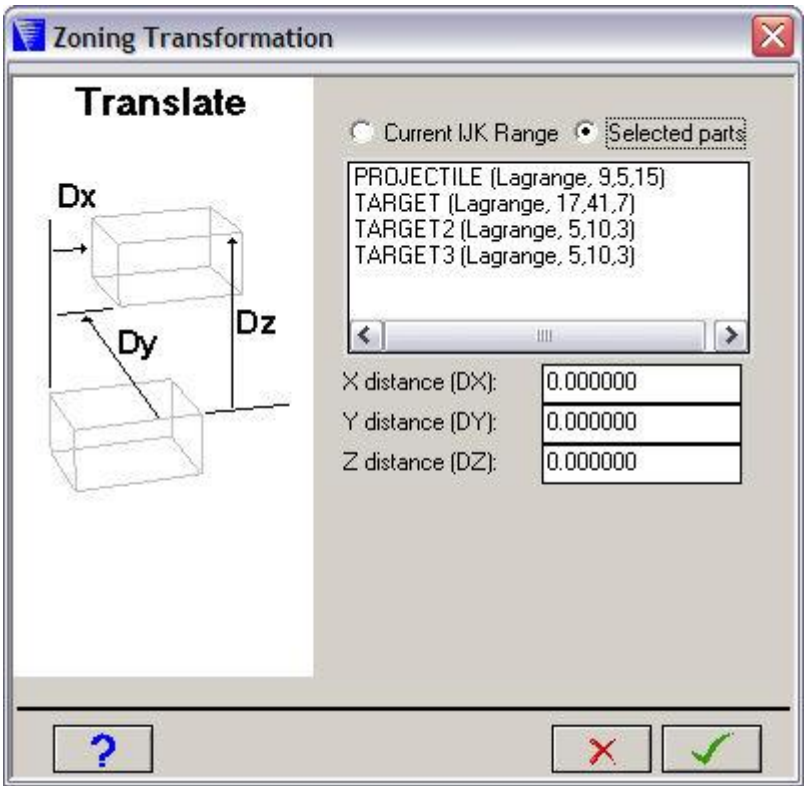
Geometric Ratio

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter a geometric ratio in this field if you want to grade the zoning instead of equally spacing the nodes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Translate



This window lets you translate nodes in XYZ space.

Current IJK Range

Select this option to apply the translation to all nodes in the current IJK Range.

Selected Parts

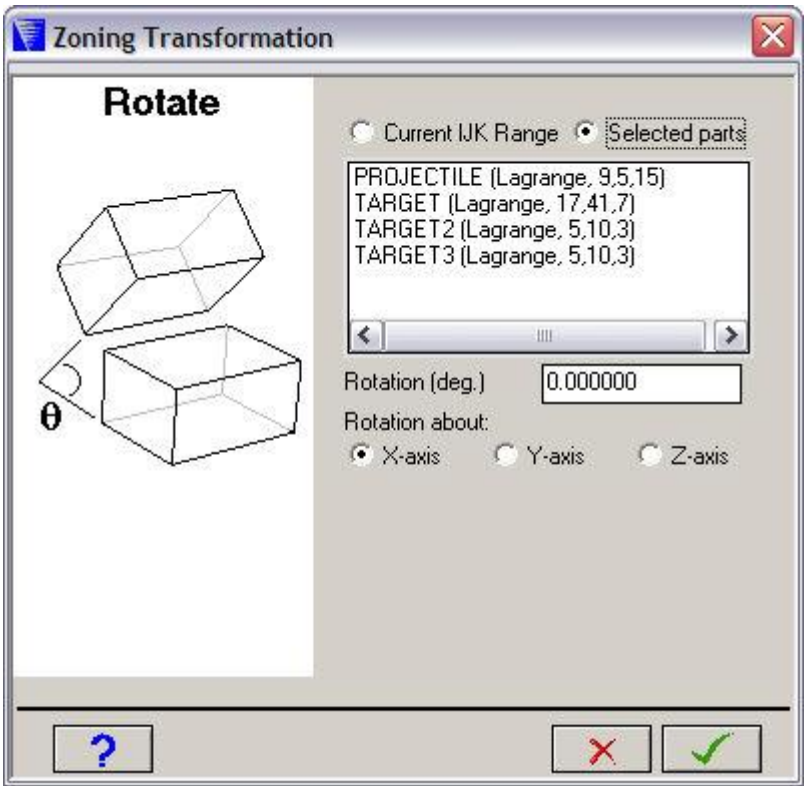
Select this option to apply the translation to all nodes of selected Parts. A box lets you select one or more Parts.

X/Y/Z distance

Input the X, Y and Z components of the translation in these fields.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Rotate



This window lets you rotate nodes about an axis in XYZ space.

Current IJK Range

Select this option to apply the rotation to all nodes in the current IJK Range.

Selected Parts

Select this option to apply the rotation to all nodes of selected Parts. A box lets you select one or more Parts.

Rotation

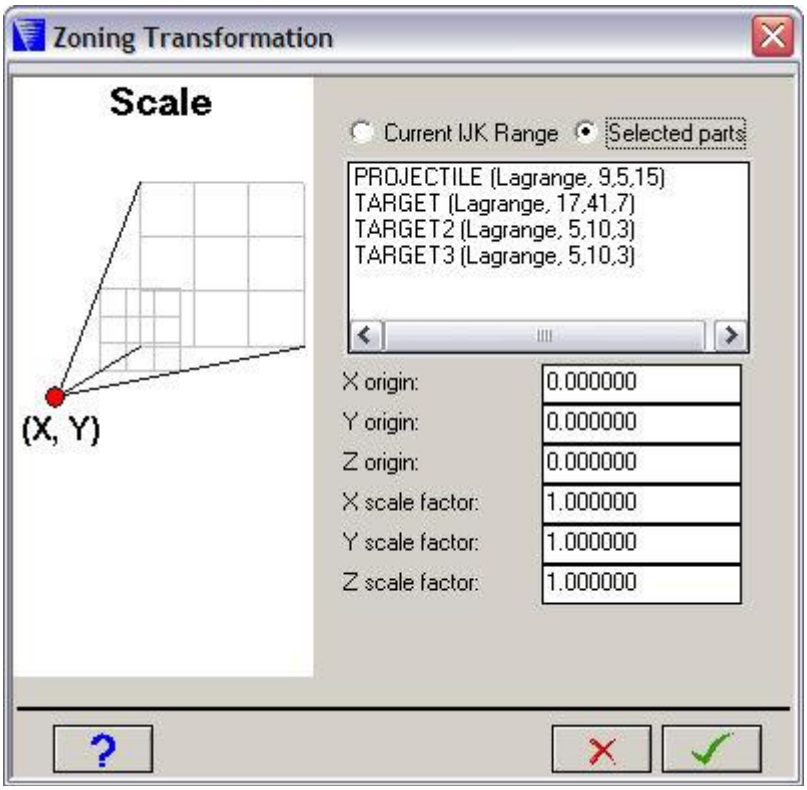
Input the rotation angle in degrees.

X/Y/Z-axis

Specify the axis about which the nodes will be rotated.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Scale



This window lets you scale nodes relative to an origin in XYZ space.

Current IJK Range

Select this option to apply the rotation to all nodes in the current IJK Range.

Selected Parts

Select this option to apply the rotation to all nodes of selected Parts. A box lets you select one or more Parts.

Origin

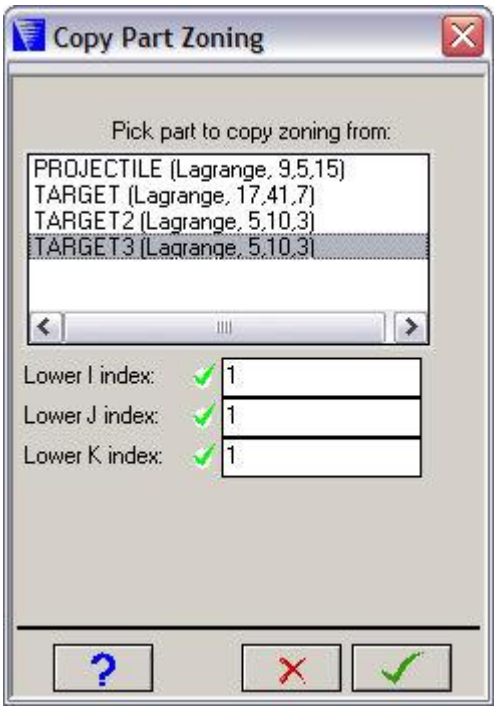
Input the X, Y and Z coordinates of the origin for the scaling.

Scale Factors

Input the X, Y and Z scale factors.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Copy



This window lets you copy zoning from an existing Part to the current IJK Range

Pick part to copy

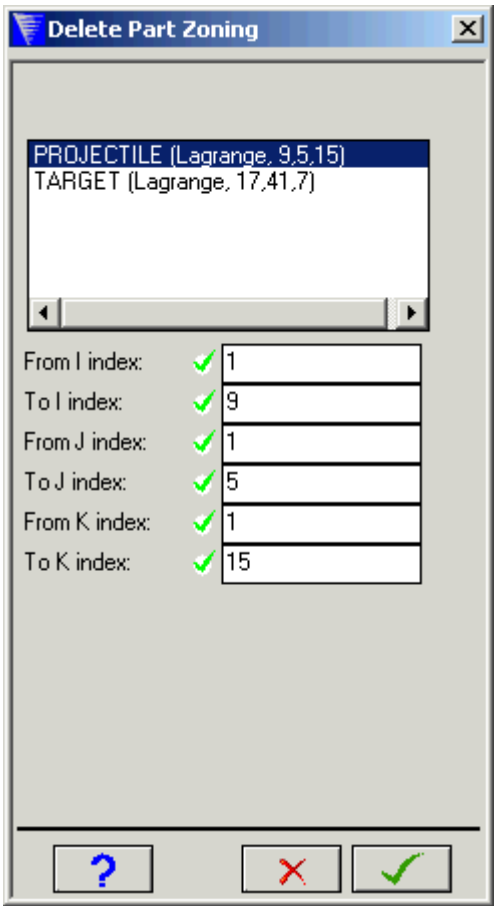
Select the Part you want copy zoning from in this box.

Lower I/J/K index

Specify the lower I, J and K indices of the block you want to copy from in these fields.
A block starting at these lower indices will be copied to the current IJK Range.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Delete



This window lets you delete the zoning in a Part.

Parts List

In the top box, select the Part in which you want to delete some zoning.

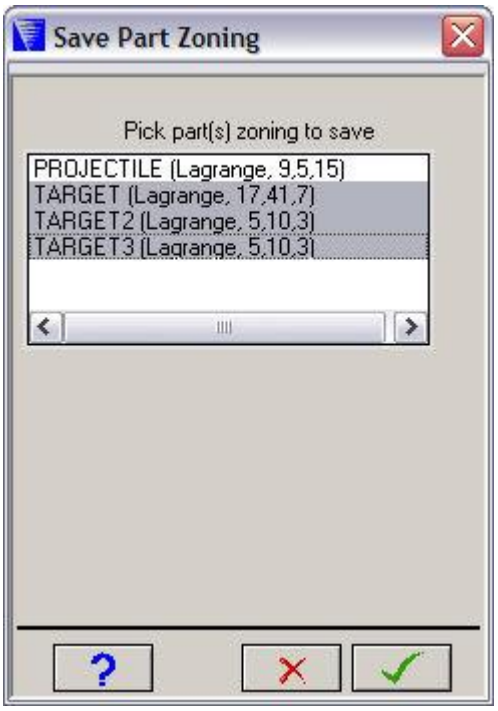
Index Range

Specify the index range of the nodes whose zoning you want to delete.

The coordinates of all nodes in the index range will be set to **Undefined** (the default state).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Save



This window lets you temporarily save the zoning for one or more Parts.

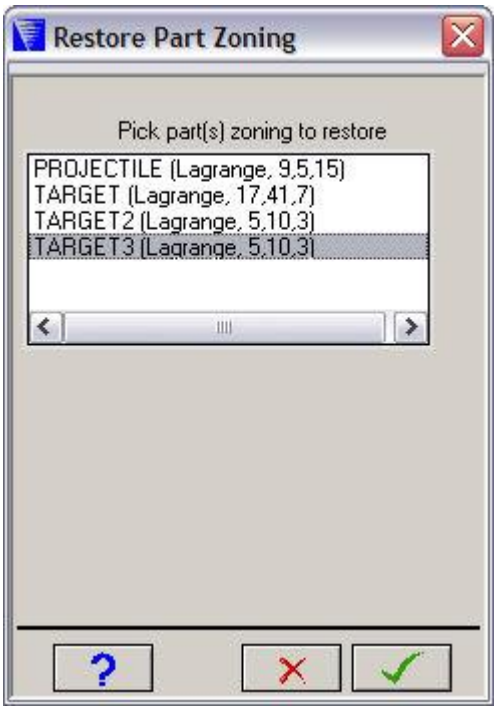
This is a useful option if you are about to do some detailed zoning changes and want to ensure you can return to your original zoning if you make a mistake.

Pressing the Restore button in the Parts-Zoning panel lets you restore the saved zoning.

Select the Parts whose zoning you want to save and press

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Restore



This window lets you restore zoning you have temporarily saved for one or more Parts.(pressing the Save button in the Parts-Zoning panel lets you temporarily save zoning for one or more Parts).

Select the Parts whose zoning you want to restore and press

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Zoning - Refine

This window lets you refine structured Lagrange, ALE, Euler-Godunov and Euler-FCT parts. However, the part must satisfy the following requirements in order to utilize this new refining facility:

- The part must be a 'box' type object.
- The X, Y and Z coordinates of the grid nodes must increase with the I, J and K indices respectively.

I/J/K Factor

Enter the Refine Factor for the I/J/K index, i.e., number of the new cells created from an original single cell in that direction.

The refining algorithms are by default applied to all cells within a part. In order to limit the range of cells within a part that undergo refinement the user can select an appropriate IJK-RANGE first. In addition it should be noted that the following limitations apply to the refinement of Euler and ALE parts.

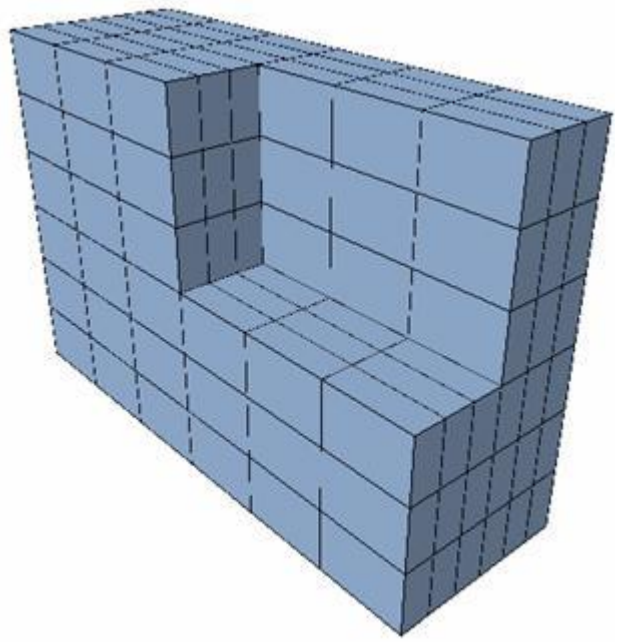
- For simplicity with multi-material Euler-Godunov grids, the material volume fraction of an original cell is assigned to all the new cells created from that original cell. Thus the material interface on the new grid may not be exactly the same as that on the original grid.

- For ALE grids, the original motion constraint of the ALE nodes is not retained during the refining procedure. The motion constraint for all the new nodes is set to Lagrange, even if the constraint of the original node is, for example, Equipotential. The user will therefore need to re-define the motion constraints for ALE parts following the refining process.

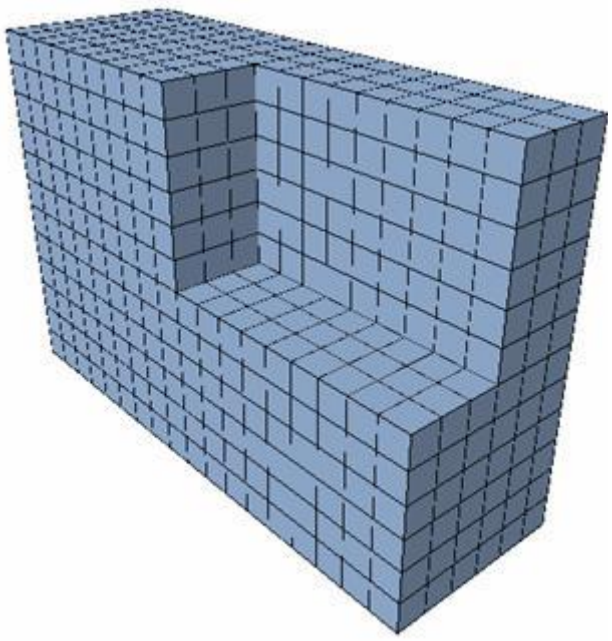
The cell index for time history gauge points is also updated during the refinement. Furthermore, the mass is always conserved between the original and refined parts. However, the momentum and kinetic energy in Lagrange and ALE parts may not be conserved exactly because the mesh refinement modifies the nodal masses and thus affects the momentum and kinetic energy calculation. For Euler-Godunov and Euler-FCT parts, since the cell mass is used to calculate both the kinetic energy and momentums, refinement does not affect the calculation. The kinetic energy and momentums should therefore always be conserved.

An example of the refinement of a Lagrangian part is shown below. Note that the refinement factors IFACT, JFACT and KFACT have been set to 1, 2 and 3 respectively.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



(a) Before refinement

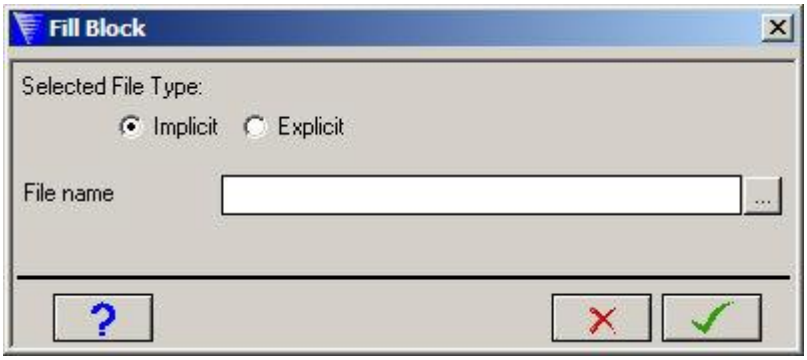


(b) After refinement

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts-Zoning-Fill Block

Parts-Zoning-Fill Block



This window lets you import complicated zoning and fill information, such as a city centre consisting of many buildings of different size. The option is only available for Euler-FCT parts.

After browsing for an input file (extension .blk), select the format of the external data.

Implicit format – AUTODYN reads the fill flag of Euler-FCT cells in a certain predefined sequence. The fill flags of ALL the cells in the input region must therefore be defined in the data file.

Explicit format – AUTODYN reads the cell index together with its fill flag. Hence, only the buildings (unused) cells and the cells with target points need to be included in the data file.

The external file imported into AUTODYN using the Fill Block option should have the extension .blk, for example block0.blk. Note that this file name is not necessarily the same as the ident of the AUTODYN model that is being pre-processed.

The Fill Block command also permits extra air space to be created beyond the input region. For example, to allow a pressure wave to propagate over the tallest building in the model and interact with the buildings behind, Fill Block can be used to create additional air space above the building.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts-Zoning-Fill Block-Implicit

IMPLICIT	Fill type
IMAX, JMAX, KMAX	Maximum I, J, K indices of input region
XMIN, YMIN, ZMIN	Minimum co-ordinates of input region (corresponding to the node at I=J=K=1
DX, DY, DZ	X, Y, Z zone dimensions for input region
XRAT, YRAT, ZRAT	Ratios used to extend grid beyond the original input region. For example, if XMAX is the maximum X-coordinate of the input region and XRAT=0.5, the generated grid would be extended to XMAX + 0.5 (XMAX-XMIN) in the X-direction, with geometric zoning being used automatically for cells above XMAX. The maximum I index of the part will be larger than the IMAX defined in the data file, hence extra elements are created to include the extended air space
SIE	Unit-dependent specific internal energy for the air (2.068×10^5 is usually OK for the mm/mg/ms units)
IFILL	Fill flag of each cell in input region (K varying faster than J, J varying faster than I. Input 20 cells per line of data IFILL= 0 Fill cell with unused IFILL= 1 Fill cell with air IFILL= 2 Fill cell with air and target point

An example of a .blk file using the implicit format is shown below. Note that the IFILL flag has to be defined for all elements in the part. The resulting filled Euler-FCT grid is shown in Figure 2.2.1.

```
IMPLICIT
6  6  6
0.0000E+00          0.0000E+00          0.0000E+00
0.1000E+03          0.1000E+03          0.1000E+03
0.4400E+00          0.4400E+00          0.4400E+00
2.0680E+05
1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1
1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   2
0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0   0
2   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1
1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1
1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1   1
1   1   1   1   1
```


Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

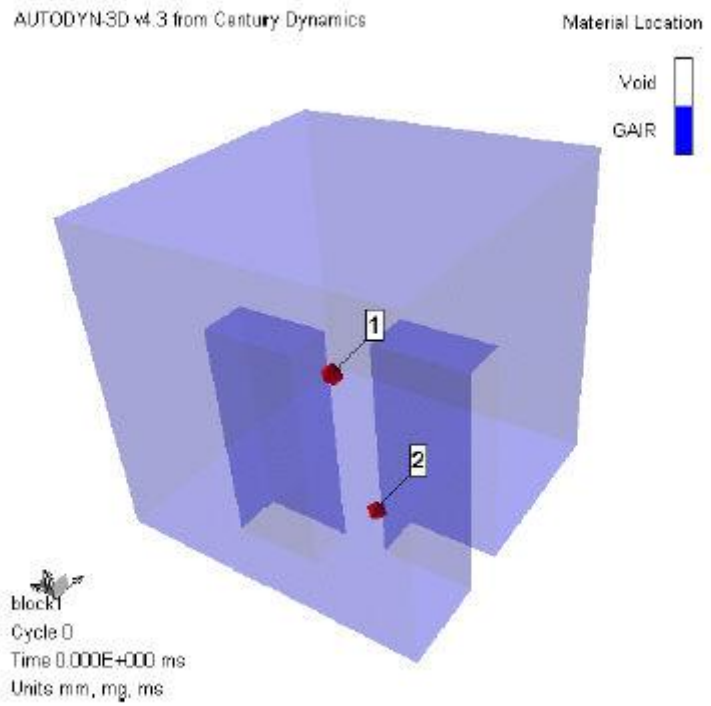


Figure 2.2.1 Example Euler-FCT grid using the Implicit Format

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts-Zoning-Fill Block-Explicit

EXPLICIT	Fill type
IMAX, JMAX, KMAX	Maximum I, J, K indices of input region
XMIN, YMIN, ZMIN	Minimum co-ordinates of input region (corresponding to the node at I=J=K=1
DX, DY, DZ	X, Y, Z zone dimensions for input region
XRAT, YRAT, ZRAT	Ratios used to extend grid beyond the original input region. For example, if XMAX is the maximum X-coordinate of the input region and XRAT=0.5, the generated grid would be extended to XMAX + 0.5 (XMAX-XMIN) in the X-direction, with geometric zoning being used automatically for cells above XMAX. The maximum I index of the part will be larger than the IMAX defined in the data file, hence extra elements are created to include the extended air space
SIE	Unit-dependent specific internal energy for the air (2.068×10^5 is usually OK for the mm/mg/ms units)
I, J, K, IFILL	I, J and K index of the cell and its fill flag. One cell per line of input IFILL = 0 Fill cell with unused IFILL = 1 Fill cell with air (default) IFILL = 2 Fill cell with air and target point

An example of a .blk file using the explicit format is shown below. Note that only elements with an IFILL flag of 0 or 2 need to be defined. The default of 1 is assumed for all other cells. The resulting filled Euler-FCT grid is shown in Figure 2.2.2.

```
EXPLICIT
11  11  11
0.0000E+00          0.0000E+00          0.0000E+00
0.1000E+03          0.1000E+03          0.1000E+03
0.5000E+00          0.5000E+00          0.5000E+00
2.0680E+05
6   6   2   0
6   6   3   0
6   6   4   0
6   6   5   0
6   6   6   0
6   6   7   0
6   6   8   0
```


Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

```
6    6    9    0
6    6    10   0
6    6    11   0
5    6    2    2
5    6    3    2
5    6    4    2
5    6    6    2
5    6    7    2
5    6    8    2
5    6    9    2
5    6    10   2
5    6    11   2
END
```

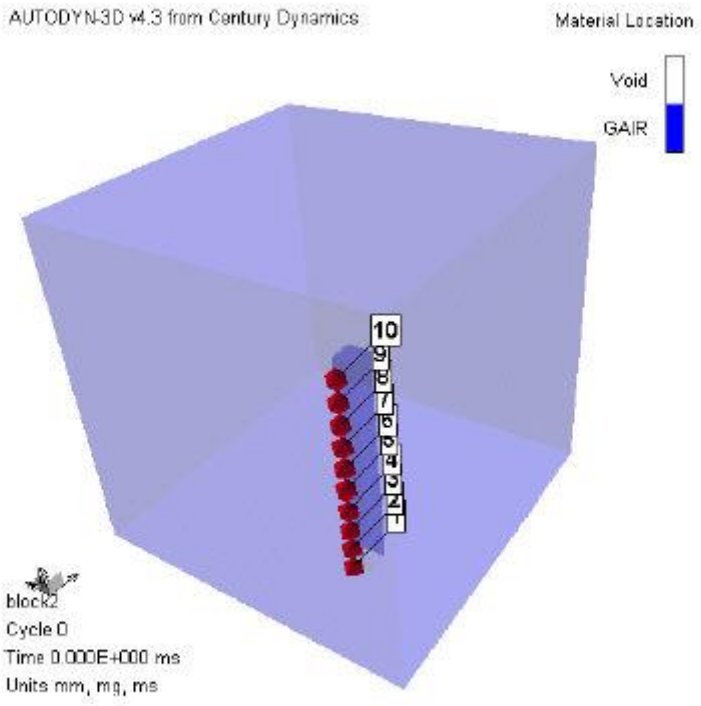


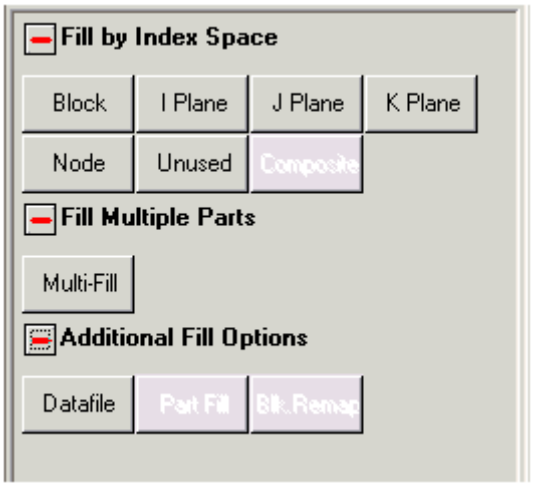
Figure 2.2.2 Example Euler-FCT grid using the Explicit Format

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

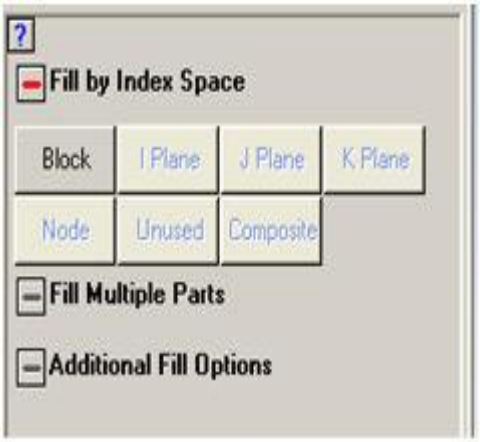
Fill

Parts - Fill

Pressing the **Fill** button in the **Parts panel** displays the following dialog in the lower region of the panel :



Structured



Unstructured

This dialog lets you fill the selected Part with materials and initial conditions.

The only option available for unstructured parts is to fill the whole part. When an unstructured parts only needs to be filled partially, the Group option can be used to create a selected group of elements which then can be filled with materials or initial conditions.

Fill by Index Space

Block
Press this button to fill a block in index space.

I/J/K Plane
Press this button to fill a plane in index space.

Node
Press this button to fill a node.

Unused
Press this button to fill a block in index space with unused elements.

Composite
Press this button to fill a composite shell.

Fill Multiple Parts

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Multi-Fill
Press this button to quickly fill multiple Parts.

Additional Fill Options

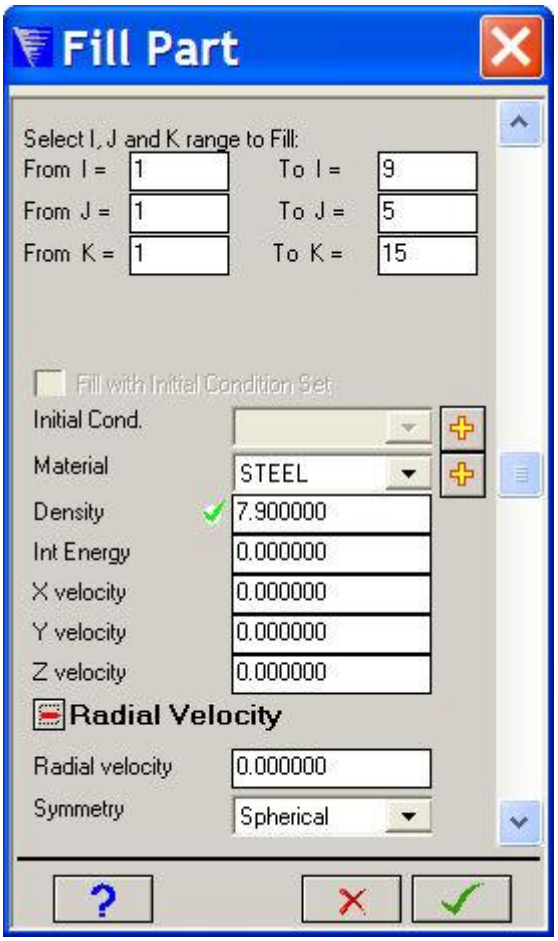
Datafile
Press this button to read or write a datafile for General Remapping.

Part Fill
Press this button to fill an Euler Part from using Lagrange or Fill Parts.

Block Remap
Press this button to read or write a datafile for Fast Block Remapping.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - Block



This window lets you fill a index block within the current Part.

Select I, J and K range to Fill

Input the I, J and K range that you want to fill in the fields provided.

Fill with Initial Condition Set

Check this box if you want to fill the block using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Material

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

You can define new materials by pressing the **Materials** button on the Navigation Bar or by pressing next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (Density, Energy, Velocity).

Radial Velocity

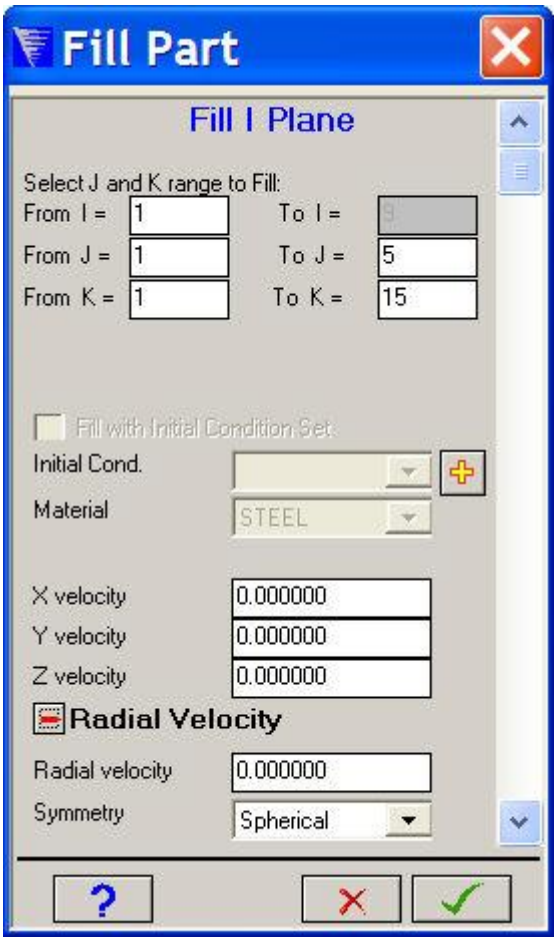
Enter the radial velocity and choose between spherical or cylindrical symmetry. For cylindrical symmetry the direction of the cylinder axis must also be specified.

Axial Velocity

Enter the axial velocity and select the origin and direction of the axis.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - I / J / K Plane



This window lets you fill a I, J or K plane of the current Part with initial conditions. The example shown is for filling a I plane (input for the other planes is similar).

Select range to Fill

Enter the I index for the plane you want to fill and the J and K range to fill in this plane.

Fill with Initial Condition Set

Check this box if you want to fill the plane using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Material

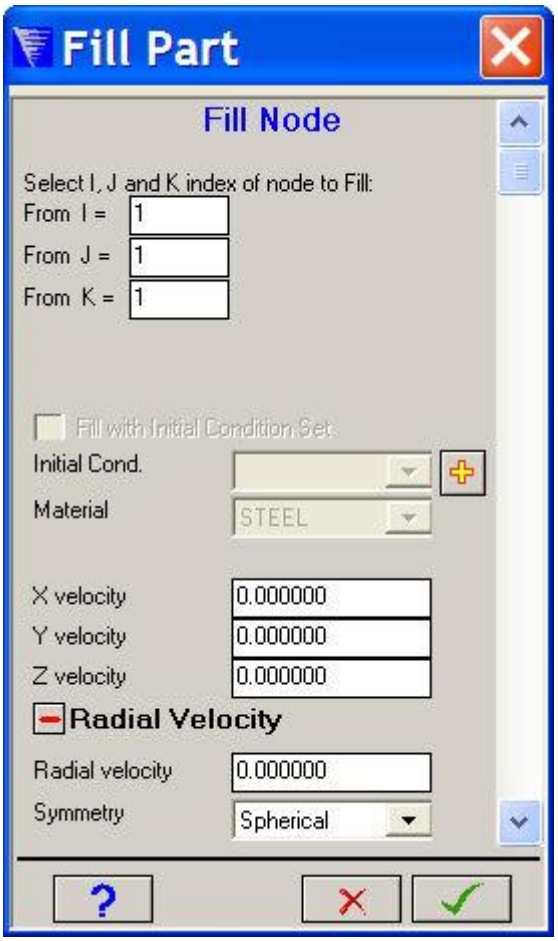
Since you are only filling a plane, no material fill is done, only initial conditions for nodes are set (i.e. velocities)

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (Velocity only).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - Node



This window lets you fill a I, J or K plane of the current Part with initial conditions. The example shows is for filling a I plane (input for the other planes is similar).

Select the node to Fill

Enter the I, J and K indices for the node you want to fill.

Fill with Initial Condition Set

Check this box if you want to fill the node using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Material

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

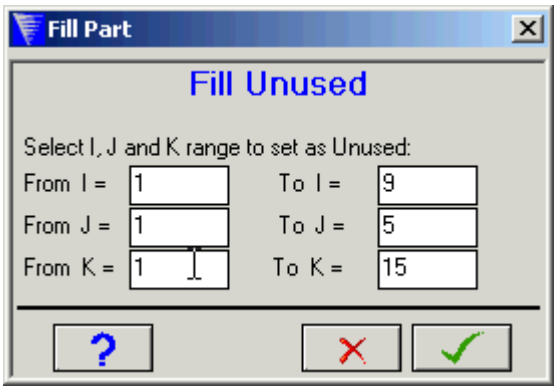
Since you are only filling a node, no material fill is done, only initial conditions for the node are set (i.e. velocities)

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (Velocity only).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - Unused



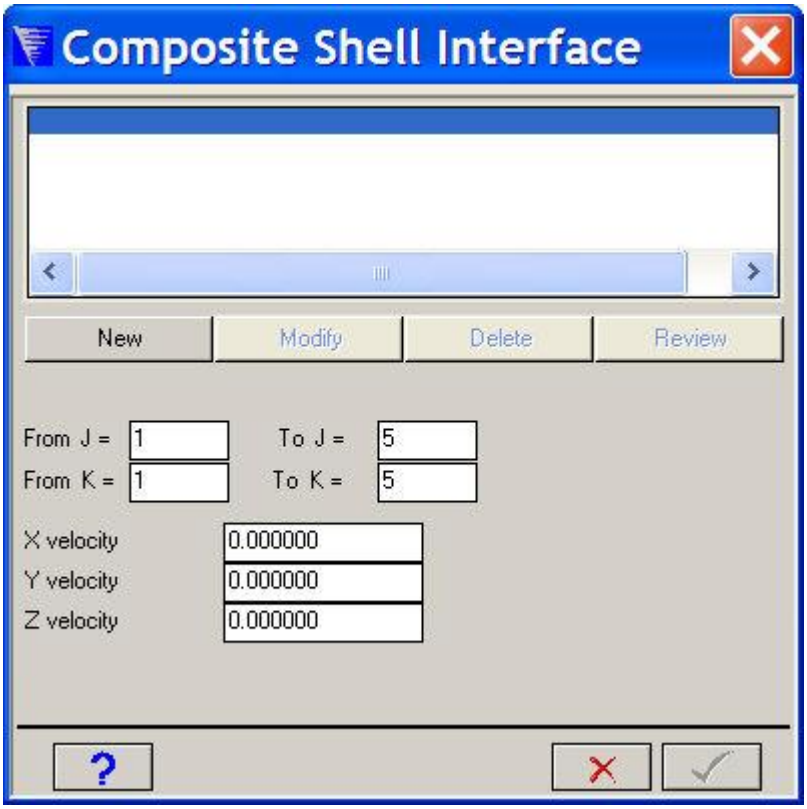
This window lets you fill a block in index space with **Unused** elements (previous materials occupying the elements are removed).

Enter the index range of the block you want to fill and press

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Composite

Parts - Fill - Composite



This window lets you define a shell as a composite shell, consisting of a number of different layers. The composite allows efficient modeling of thin composite materials which are subjected to structural, rather than shock type loading.

New

Press this button to create a new shell layer definition.

Modify

Press this button to modify an existing shell layer definition.

Delete

Press this button to delete an existing shell layer definition.

Review

Press this button to review the existing shell layer definition.

Range

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

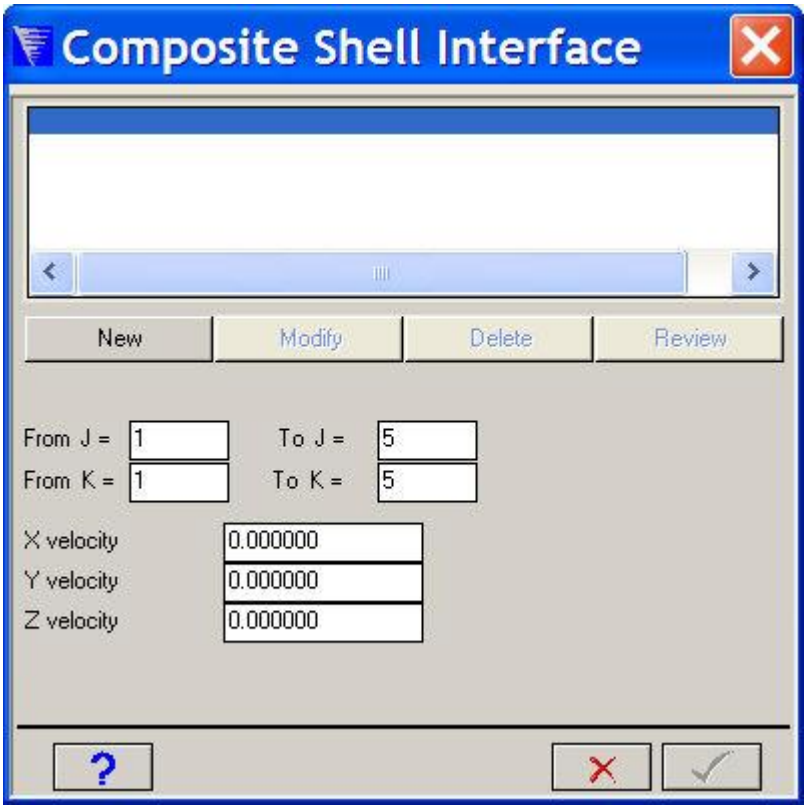
Enter the J and K range of the shell to apply the selected layer definition.

Velocity

Enter the initial velocity of the shell.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - Composite - New



This window lets you create an existing layer definition.

Number of Layers

Select the number of layers for the composite layer definition.

Layer definition name

Enter a name for the layer definition.

Material

Select the material for each material layer.

Direction

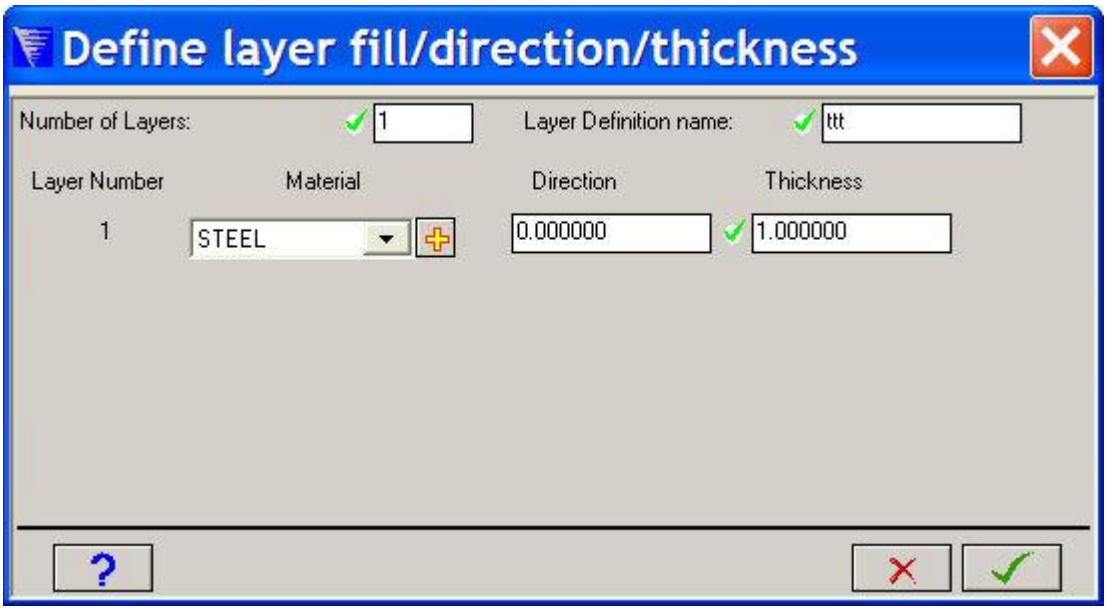
For orthotropic materials, enter a material direction in degrees.

Thickness

Enter the thickness for each material layer.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - Composite - Modify



This window lets you modify an existing layer definition.

Number of Layers

Select the number of layers for the composite layer definition.

Layer definition name

Enter a name for the layer definition.

Material

Select the material for each material layer.

Direction

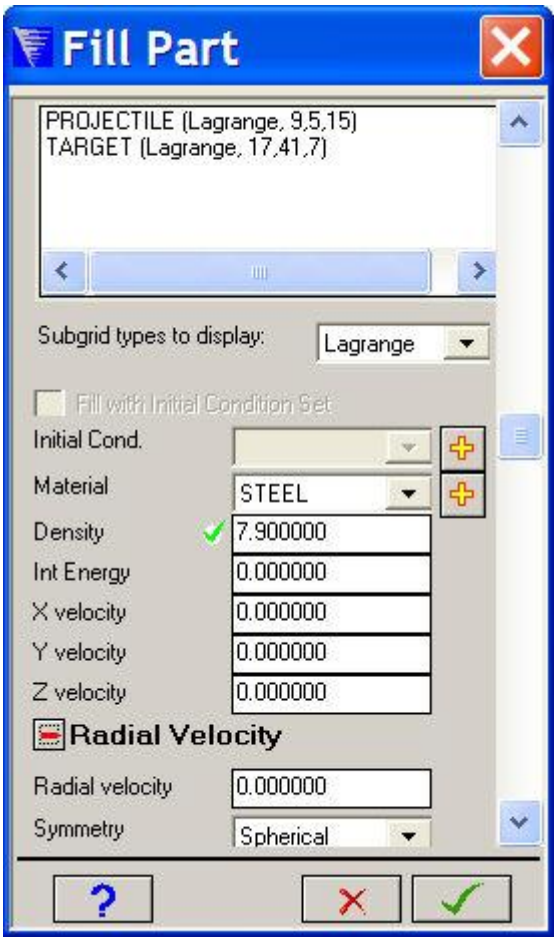
For orthotropic materials, enter a material direction in degrees.

Thickness

Enter the thickness for each material layer.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - Multi-Fill



This window lets you quickly fill multiple Parts. Each selected Part must be filled in its entirety.

Fill Parts

The top box displays a list of Parts that you can select to fill.

Below this box is a pull-down menu that lets you choose the type of Parts from which you want to make your selection (you can only fill Parts of the same type at the same time).

Fill with Initial Condition Set

Check this box if you want to fill the Parts using an Initial Condition Set.

The advantage of filling a Part using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Material

If you do not want to use an Initial Condition Set, select the material you want to use to fill your Part from this Pull-down menu.

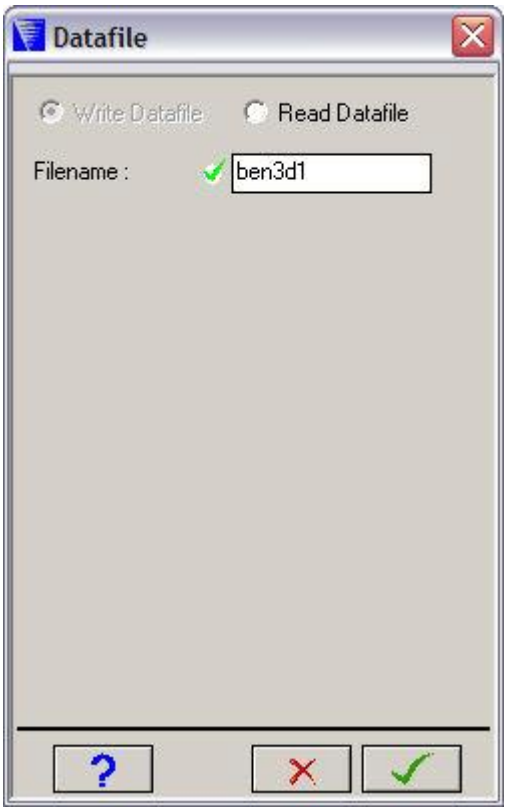
You can define new materials by pressing the **Materials** button on the Navigation Bar or by pressing next to the option in this window.

Parameters

If you do not want to use an Initial Condition Set, enter parameter values for your fill in the fields provided (Density, Energy, Velocity).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

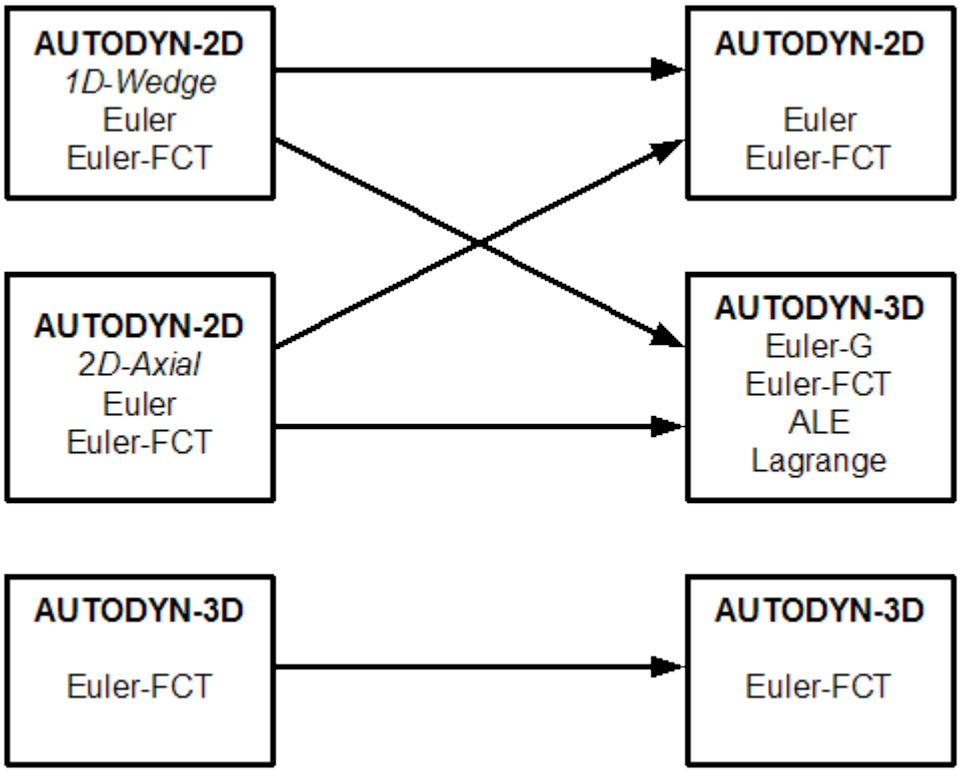
Parts - Fill - Datafile



This window lets you write fill data to or from datafiles (**.fil**)

These data files are commonly used for remapping solutions from 1D Wedge, 2D axysymmetric and 3D models into other 2D / 3D models. The figure below illustrates the remap operations available in AUTODYN

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



- The AUTODYN-2D remap (data) file writing capability includes all state variables including the stress tensor etc. You retain the existing option to only write hydrodynamic data if you wish to keep the size of the remap files to a minimum.

The AUTODYN-2D remap (data) file import capability allows 2D to 2D remapping of all state variables including the stress tensor etc.

- The AUTODYN-3D remap (data) file import capability includes all state variables including the stress tensor etc.

- In AUTODYN-3D an approximate remap into single material ALE and Lagrange solvers has also been included, that is cells are filled with the material with the largest volume fraction. If this happens to be void material then the element is set to unused.

- When reading remap data files, the origin of the remap can be user defined in both AUTODYN-2D and 3D. Furthermore, in AUTODYN-3D the axis of symmetry for the remap can be aligned with either the X(I), Y(J) or Z(K) direction. This reduces the existing constraint that the X-axis needed to be consistent between the 2D and 3D models whilst keeping the geometry calculation simple.

- Note that all remap data files created in older versions of the software can still be read into the latest version AUTODYN, although they will only contain hydrodynamic material data.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

- The results of the remapping of the Johnson-Holmquist material model cannot be guaranteed to be accurate when the material is damaged considerably, since voids are artificially closed during remapping. The user should therefore take care to check results carefully when using the Johnson-Holmquist material model in the remapping procedure.

Write Datafile

Select this option if you want to write fill data for the current Part to a **.fil** file.

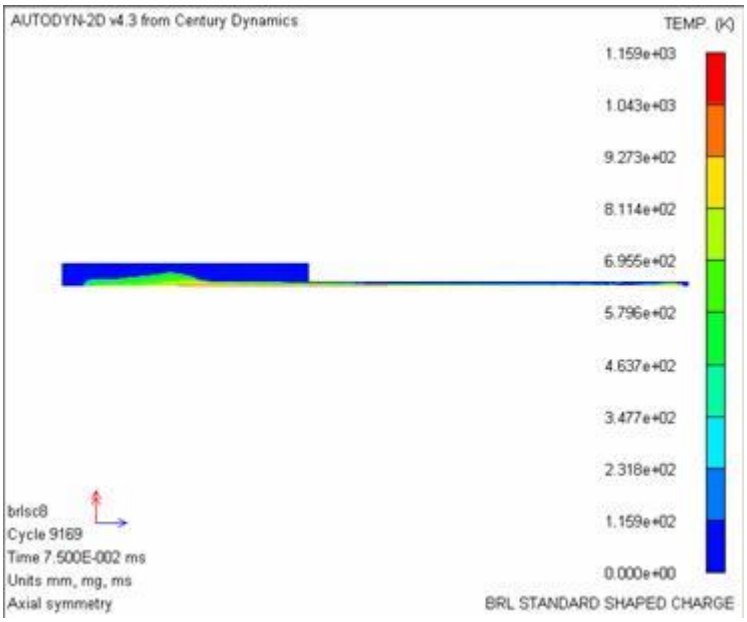
Read Datafile

Select this option if you want to read fill data for the current Part from a previously written **.fil** file.

Filename

Enter the file name in this field (do not append **.fil**)

An example of a shaped charge analysis is shown in the below. The formation of the shaped charge jet is first modeled using the multi-material Euler solver in AUTODYN-2D, as shown below. A remap data file is created at the desired wrapup time.

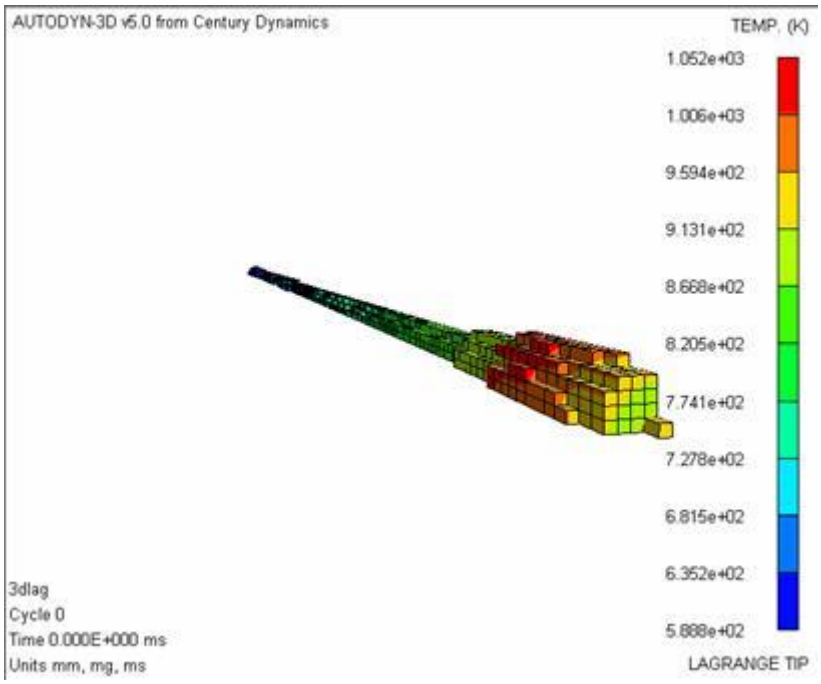


Formation of shape charge jet in AUTODYN-2D

The analysis can then be continued using AUTODYN-3D. In this example a Lagrange part is first created and zoned in order that the part will completely enclose the volume occupied by the shaped charge jet. At this stage the part is not filled and all cells are left as unused. Note that it is recommended that cell sizes are chosen such that at least 4 filled elements will exist across the radius of the shaped charge jet. These cell sizes can be determined by examining the geometry at the wrapup cycle of the AUTODYN-2D analysis.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

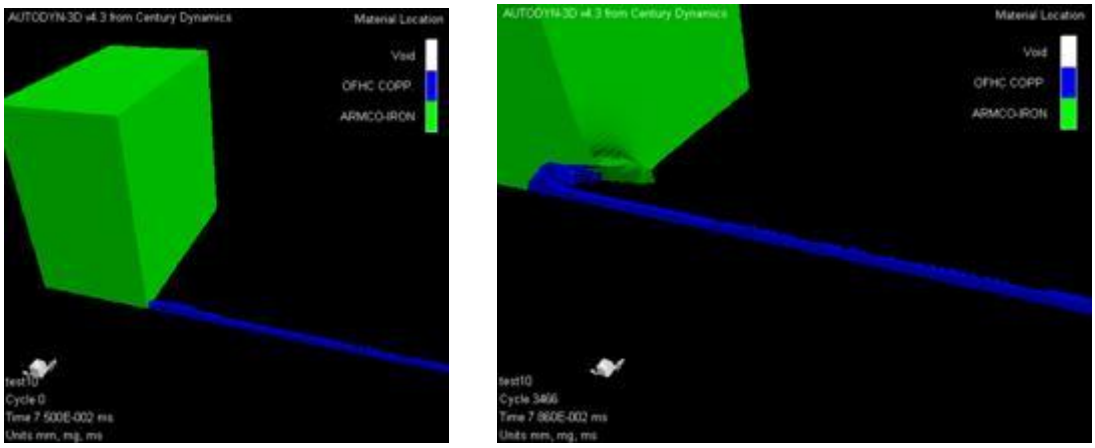
In AUTODYN-3D the results from the 2D multi-material Euler analysis are remapped into the empty 3D Lagrangian part. The Lagrangian cells that contain more than 50% material following the remap are set as used cells. The profile of the Lagrangian jet will therefore not exactly match that of the 2D case, although accuracy can be enhanced by using the greatest level of refinement that computation time constraints will allow. Note that the constitutive state variables and velocities for the used cells are also remapped into the 3D analysis. An example of the tip of a shaped charge jet following the remap procedure described above is shown below.



Lagrangian jet following remap into AUTODYN-3D

The 3D Lagrangian part can then be translated or rotated and used in several analyses including normal and oblique impacts onto several different targets. The following figures show an example in which a Lagrangian target has been introduced into the analysis and the penetration of the jet into the target has been simulated. Note that erosion models were included for both the jet and target materials.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



(a) **(b)**

Normal impact onto Lagrangian target in AUTODYN-3D
(a) initial configuration,(b) shape charge penetrates target

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - Part Fill

This window lets you fill an Euler Part by mapping the materials and initial conditions of an existing structured **Lagrange** or **Fill** Part or unstructured volume-element parts into the Euler Part.

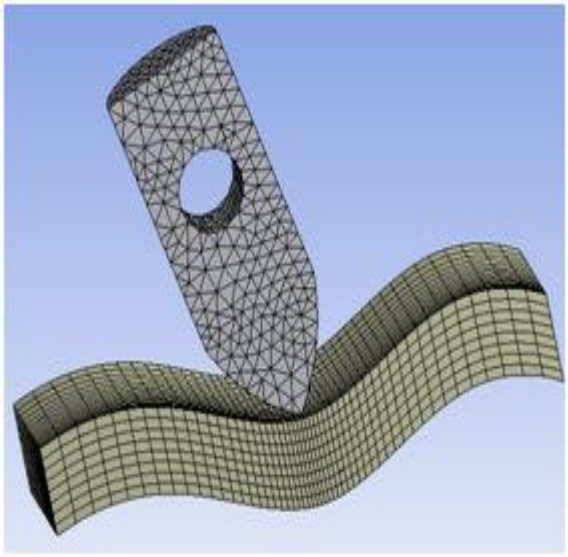
Select Part to fill into current Part

In this box, select the Lagrange or Fill Part you want to use to fill your current Euler Part.

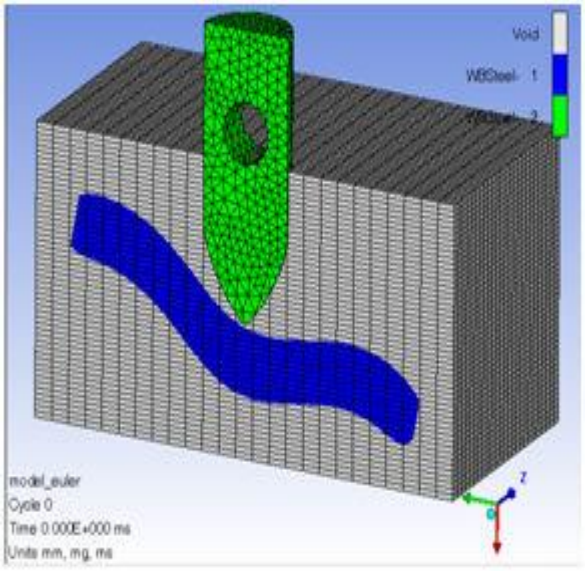
Material to be replaced

Select the material to be replaced during the fill operation (this information is only used if Euler cells contain multiple materials).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



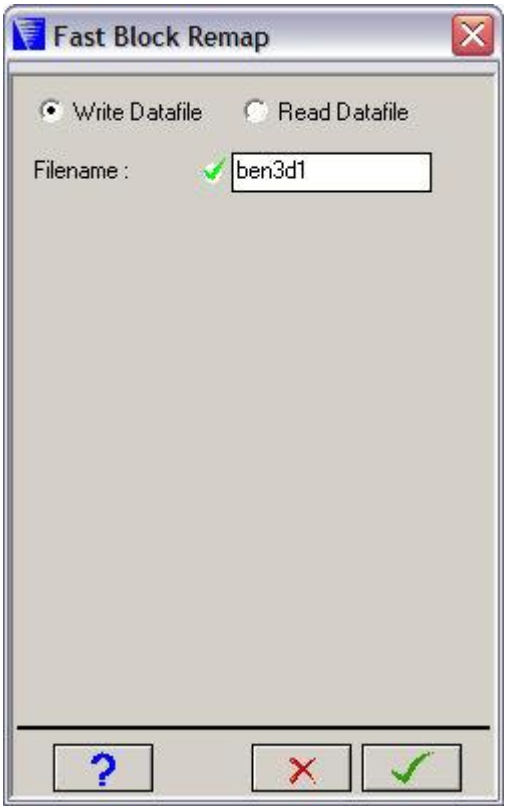
Mesh of Projectile and Target



Multi-material mesh filled with Unstructured Part

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Fill - Block Remap



This window lets you write fill data to or from datafiles (**.fil**) for performing fast block remaps.

The standard remapping scheme for 3D to 3D Euler-FCT is very useful in blast modeling but in certain circumstances can take a very long time to search for the cells in the remapping range, especially if the part size is very large, for example more than 2 million cells.

The "fast block" remapping command maps several cells from a fine mesh of a small space into a single cell in a relatively coarse mesh of a large space, directly without searching for the remapping range. Thus the efficiency of the remapping process has been improved dramatically.

The fast block remapping procedure is applicable to Euler-FCT parts only. The scheme allows unused cells in the remapping region. However, cover fractions are not considered in the remapping calculations so block remapping should not be applied to any region of the part where Euler-FCT/Lagrange or Euler-FCT/Shell coupling takes place.

Write Datafile

Select this option if you want to write fill data for the current Part to a **.fil** file.

Read Datafile

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select this option if you want to read fill data for the current Part from a previously written **.fil** file.

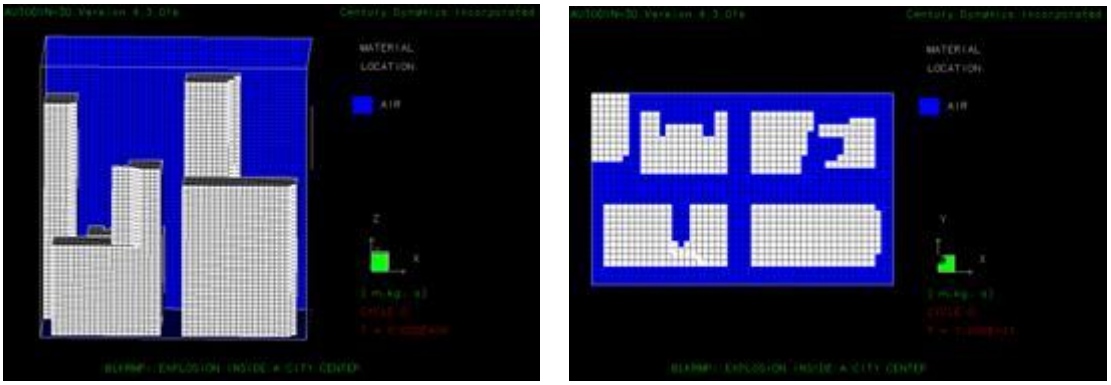
Filename

Enter the file name in this field (do not append **.fil**)

Typical steps that will be undertaken when using Block Remapping for blast problems are as follows:

1. Load the original building data file.
2. Modify the part so it only contains a small area of the explosive and the surrounding buildings.
3. Refine the mesh, then remap 1D or 2D data to the refined part.
4. Save the file and start calculation.
5. At the wrapup cycle (when the explosive wave approaches the boundary of the refined part), output the blast field to a remap file.
6. Load the original building data file again.
7. Modify the part so it contains a larger area.
8. Refine the mesh.
9. Read the remap file.
10. Save the file and continue running.
11. Repeat 5-10 as necessary.

An example of an analysis of a blast in a city centre that was performed using the block remapping facility is shown below. The city centre is represented using an Euler-FCT mesh with unused regions to represent buildings as shown below. Note that the original zoning and fill operations for this model were set up using the new block fill command.

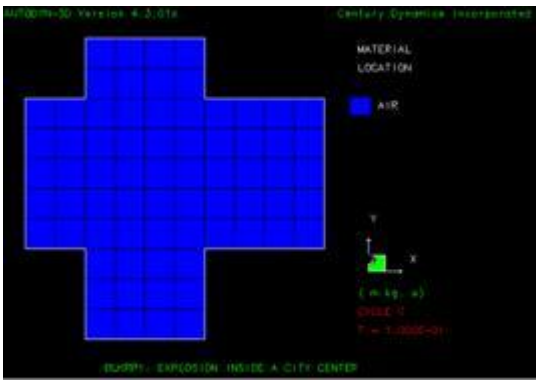


(a) side view (b) plan view

Layout of original Euler-FCT part representing city centre

To perform the initial expansion of the blast the central section of the grid, as shown below in figure a, is taken and refined by a factor of 4 as shown in figure b.

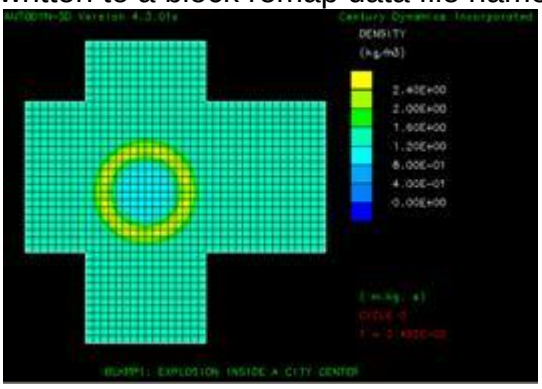
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

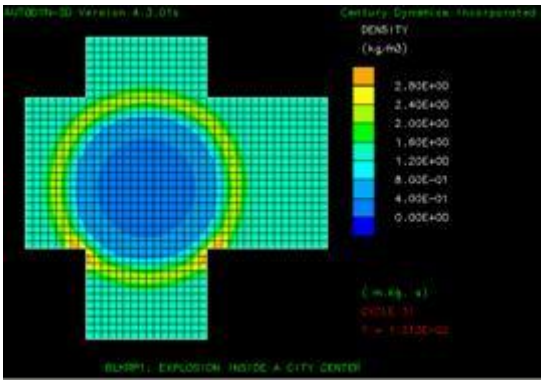


(a) unrefined (b) refined by factor of 4x4x4

Central section of part (Range I=21-31, J=11-21, K=1-21)

The analysis of the detonation of the spherical charge is initially performed using a 1D wedge analysis which is then remapped into the refined grid as shown below. The expansion of the blast wave in the central region of the city is then performed using the refined 3D Euler-FCT grid until the wave front reaches the boundaries of the refined region as shown in figure b. At this point the results for this analysis are written to a block remap data file named BLKRP1.





(a) (b)

(a) initial blast remap from 1D wedge data
(b) blast wave reaches boundary of refined region

Expansion of blast in refined central region

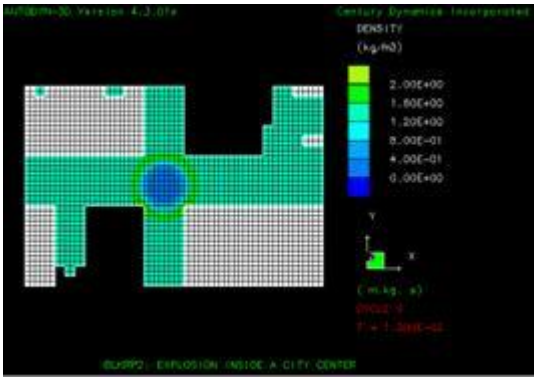
Following the creation of the remap data file BLKRP1 the original model is subsequently reloaded and a larger region of the Euler-FCT part is analyzed as shown below in figure a. This section is refined by a factor of 2 in all directions to give the resulting mesh shown in figure b.

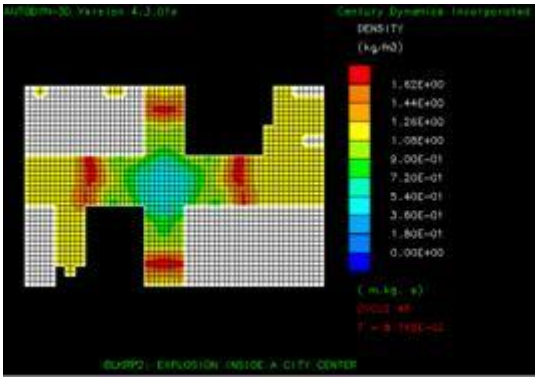
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

(a) unrefined (b) refined by factor of 2x2x2

Larger central section of part (Range I=11-41, J=6-26, K=1-41)

The results stored in file BLKRP1 are then remapped into the refined grid to give the initial conditions for the second phase of the analysis as shown below in figure a. The propagation of the blast wave can then continue until again the wave front approaches the boundaries of the refined region as shown in figure b. At this stage a further block remap data file is written named BLKRP2.





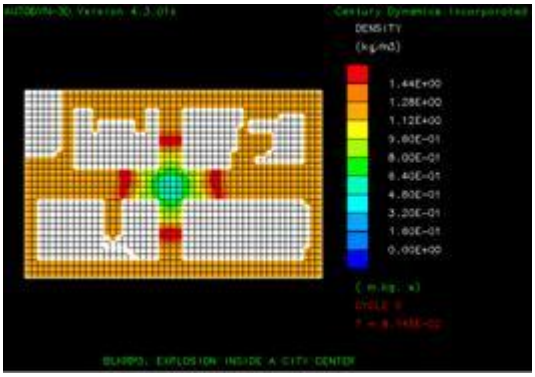
(a) (b)

(a) initial blast remap from BLKRP1
(b) blast wave approaches boundary of refined region

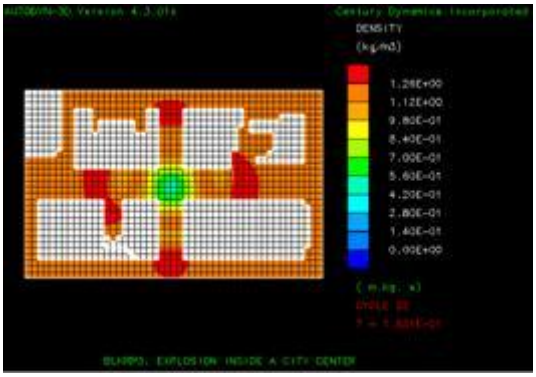
Expansion of blast in large refined region

The final stage of the analysis can be performed using the full model. This model is reloaded and the results stored in file BLKRP2 are remapped into the entire Euler-FCT part to give the initial conditions for the final phase of the analysis as shown below in figure a. The propagation of the blast wave can then be tracked though the remainder of the city centre as shown in figure b.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



(a)



(b)

(a) second blast remap from BLKRP2
(b) blast wave propagates through city centre

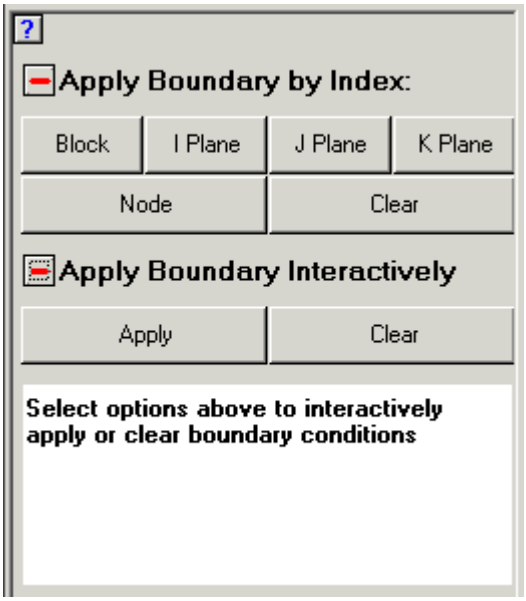
Expansion of blast in full Euler-FCT part

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

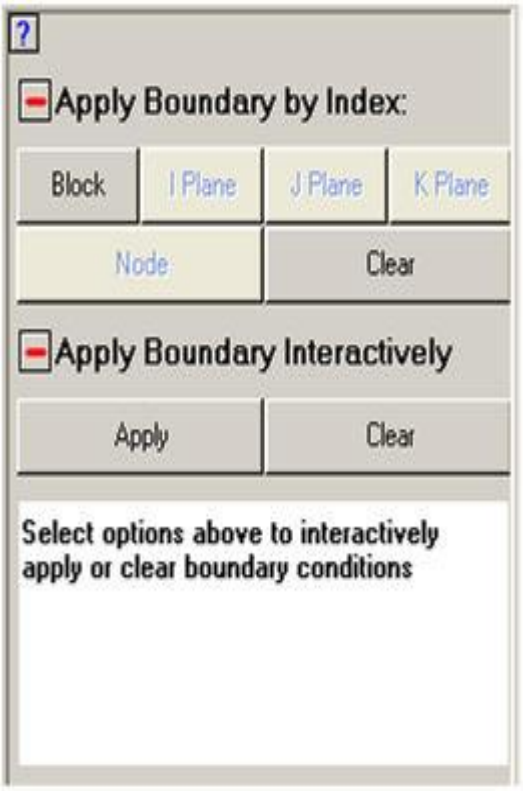
Boundary

Parts - Boundary

Pressing the **Boundary** button in the **Parts panel** displays the following dialog in the lower region of the panel :



Structured



Unstructured

This dialog lets you apply boundary conditions to the selected Part.

Block
Press this button to apply a boundary condition to a block in index space.

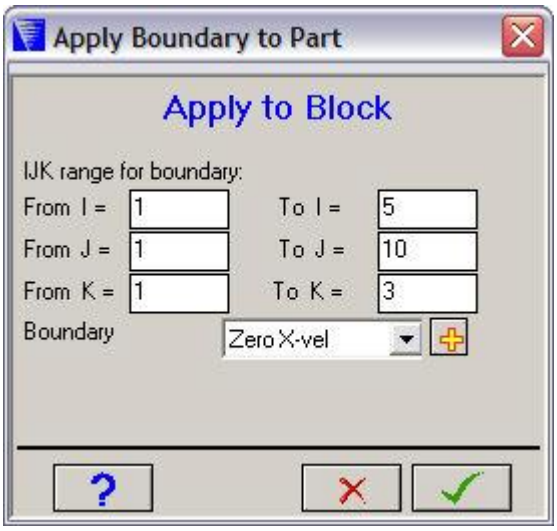
I/J/K Plane
Press this button to apply a boundary condition to a plane in index space.

Node
Press this button to apply a boundary condition to a node.

Clear
Press this button to clear previously applied boundary conditions.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Boundary - Block



This window lets you apply a boundary condition to a block in index space of the current structured Part.

For an unstructured Part the boundary condition will be applied to all nodes, elements or faces of a Part.

IJK range for boundary

Enter the IJK range of the block in the fields provided.

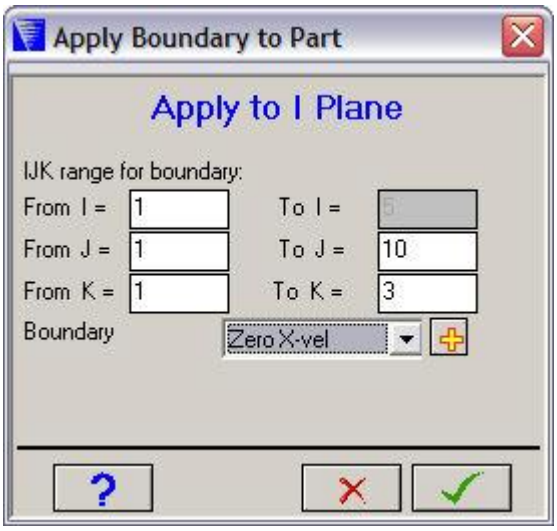
Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by pressing the **Boundaries** button in the Navigation Bar, or you can press next to the pull-down menu.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Boundary - I / J / K Plane



This window lets you apply a Boundary Condition to a I, J or K plane of the current Part.
The example shown is for applying a Boundary Condition to a I plane (input for the other planes is similar).

IJK range for boundary

Enter the I plane and the J and K range in the fields provided.

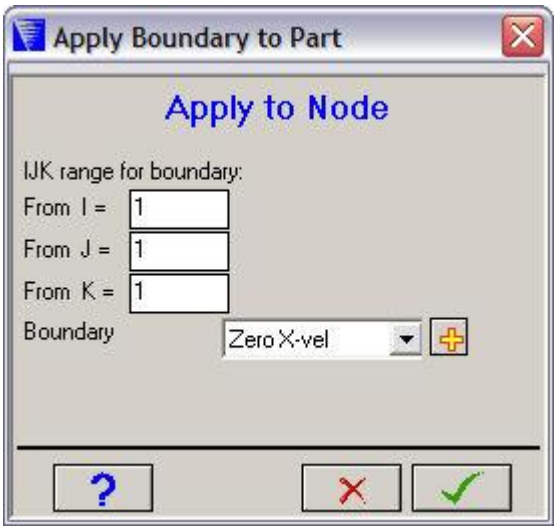
Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by pressing the **Boundaries** button in the Navigation Bar, or you can press next to the pull-down menu.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Boundary - Node



This window lets you apply a Boundary Condition to a node.

IJK range for boundary

Enter the I, J and K index for the node in the fields provided.

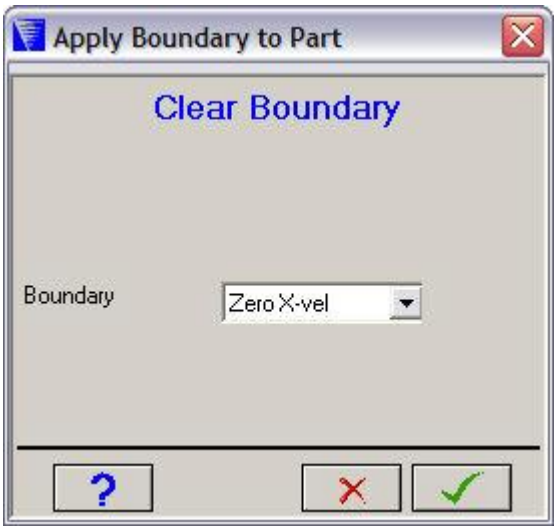
Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by pressing the **Boundaries** button in the Navigation Bar, or you can press next to the pull-down menu.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Boundary - Clear



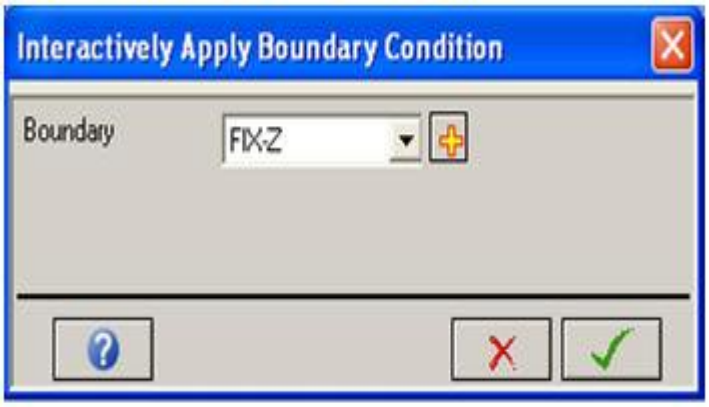
This window lets you clear a Boundary Condition from the current Part.

This operation does not delete the definition of the Boundary Condition, so you can re-apply it later.

Select the boundary condition you want to clear from the pull-down menu and press

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Boundary - Apply Interactively



This window lets you apply a boundary condition interactively by creating a 2D polygon in the graphics display. The boundary condition is applied to any nodes or faces that lie within the 2D polygon.

Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by pressing the **Boundaries** button in the navigation bar, or you can press next to the pull-down menu.

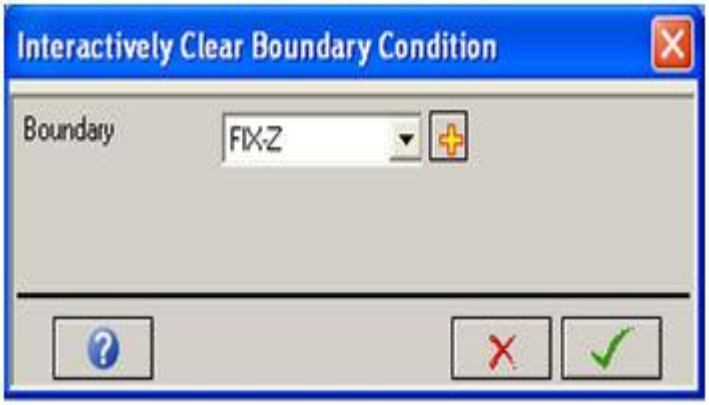
Close the window with the button, to enter the polygon definition stage.

Select the polygon points using the Alt key together with the left mouse button. To delete the last polygon, use the Shift button together with the left mouse button. To complete the polygon, use the Control and left mouse button.

After completing the polygon, the selected nodes or faces are displayed. Press the button to accept this selection, and apply the boundary condition. At any stage, the button may be used to cancel the process.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Boundary - Clear Interactively



This window lets you clear a boundary condition interactively by creating a 2D polygon in the graphics display. The boundary condition is removed from any nodes or faces that lie within the 2D polygon.

Boundary

Select the boundary condition you want to clear from this pull-down menu.

Close the window with the button, to enter the polygon definition stage.

Select the polygon using the Alt key together with the left mouse button. To delete the last polygon point, use the Shift button together with the left mouse button. To complete the polygon, use the Control and left mouse button.

After completing the polygon, the selected nodes or faces are displayed. Press the button to accept this selection, and clear the boundary condition. At any stage, the button may be used to cancel the process.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Gauges

Parts - Gauges

Pressing the **Gauges** button in the **Parts panel** displays the following dialog in the lower region of the panel :

This dialog lets you define gauge points in the current Part.

Add
Press this button to add one or more gauge points.

Move
Press this button to relocate a gauge point.

Delete
Press this button to delete one or more gauge points.

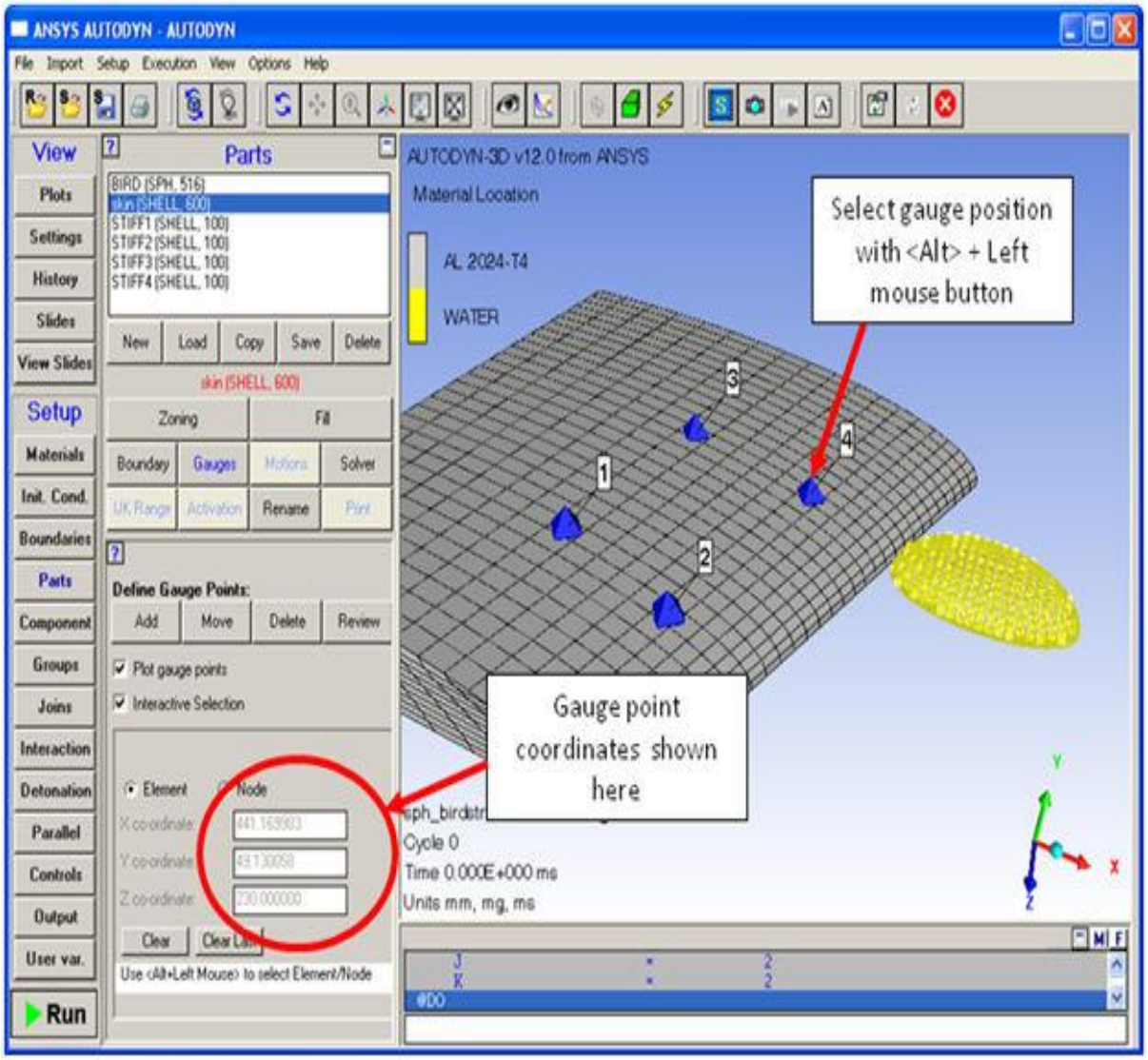
Review
Press this button to review all your gauge points in a window.

Plot gauge points
Check this box to see the location of you gauge points in the view panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactive Selection

Use the mouse to interactively select Elements or Nodes where you wish to place your gauge points by switching on Interactive Gauge points in the Gauges Panel.



The IJK indices or XYZ position of the selected element/node will be displayed, as they are selected.

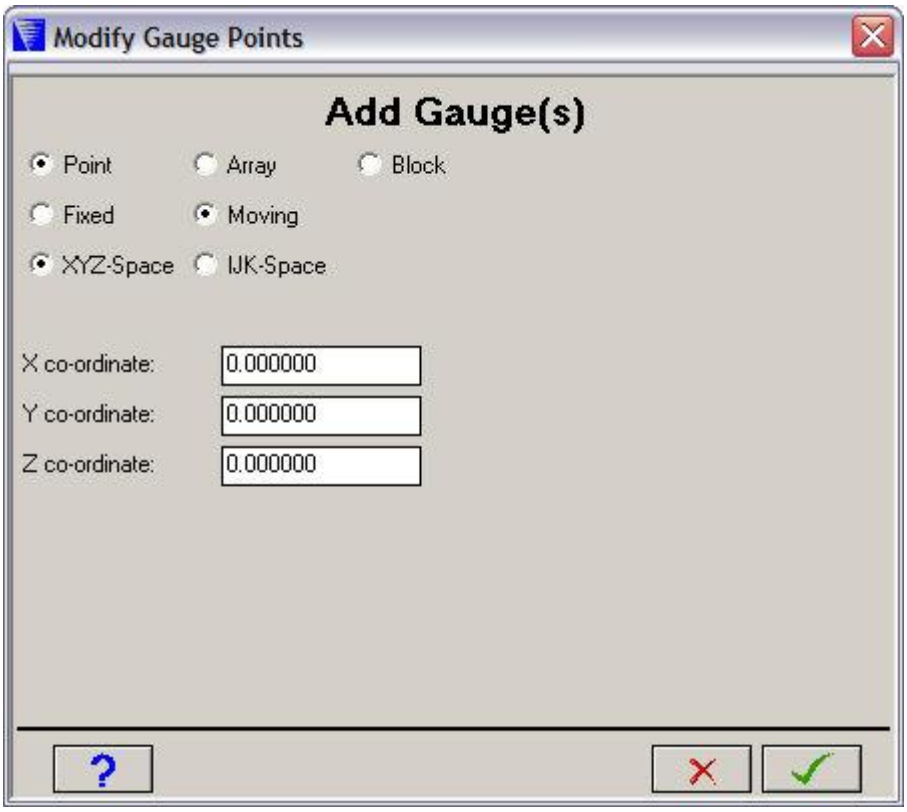
Gauges added via this method will also be cached until cleared by selecting the Undo button, or the Gauges Panel is closed.

When adding Gauges via this method the part containing the Element or Node selected becomes the selected Part in the Parts panel.

The interactive selection of Gauge Points is not available in examine mode, and the panel will not be visible if this is the case.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Gauges - Add



This window lets you add gauges to your model.

A gauge is a location where time histories of selected variables are recorded during a calculation.

Point / Array / Block

Select whether you want to define a single gauge (**Point**), a 2D array of gauges (**Array**) or a 3D block of gauges (**Block**).

Fixed / Moving

Select whether you want the gauge(s) to be fixed in space or move with the flow (as if imbedded in the material).

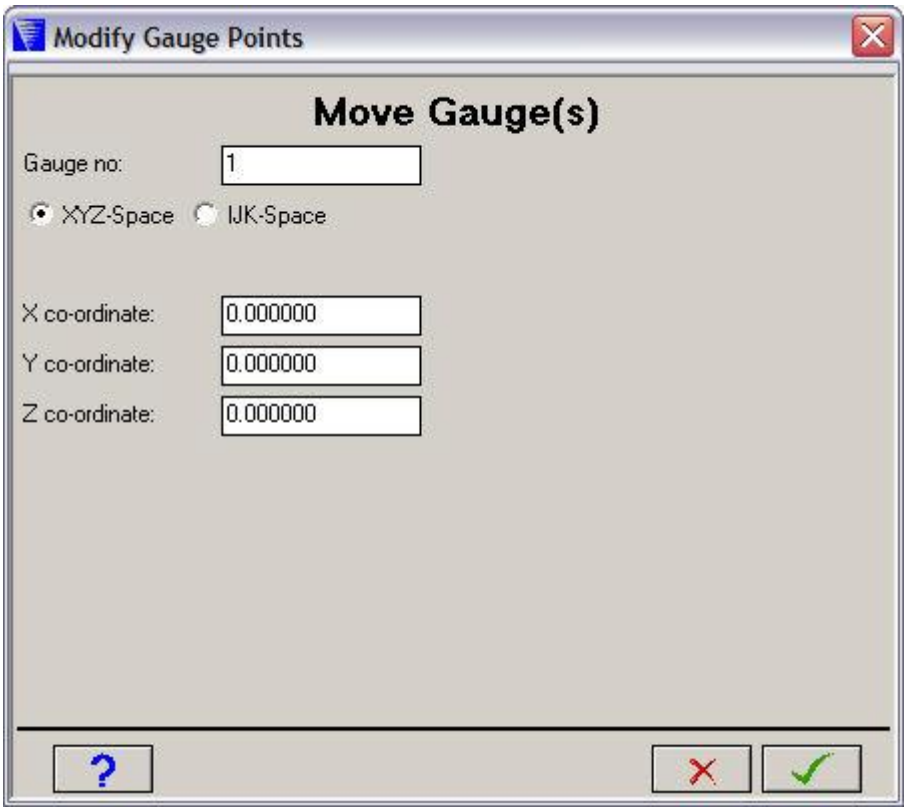
XYZ / IJK Space

Select whether you want to specify your gauge locations in real space (XYZ) or index space (IJK)

Depending on your selections, the remaining fields will ask you to specify the (X,Y,Z) coordinates of your gauges or the (I,J,K) indices of the elements in which your gauges are to be placed.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Gauges - Move



This window lets you relocate one gauge point.

Gauge Number

Enter the number of the gauge you want to relocate in this field (gauges are assigned numbers in the order in which you define them).

XYZ / IJK Space

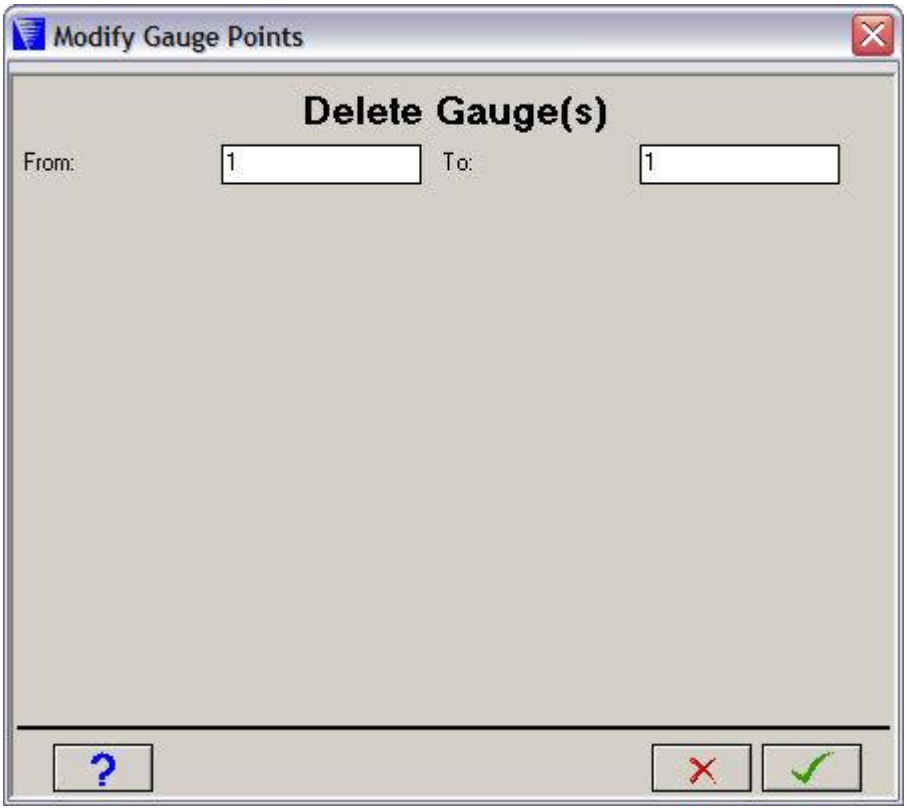
Select whether you want to specify your gauge locations in real space (XYZ) or index space (IJK)

New Position

Enter the (X,Y,Z) coordinates or (I,J,K) indices for the new position of the gauge.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Gauges - Delete

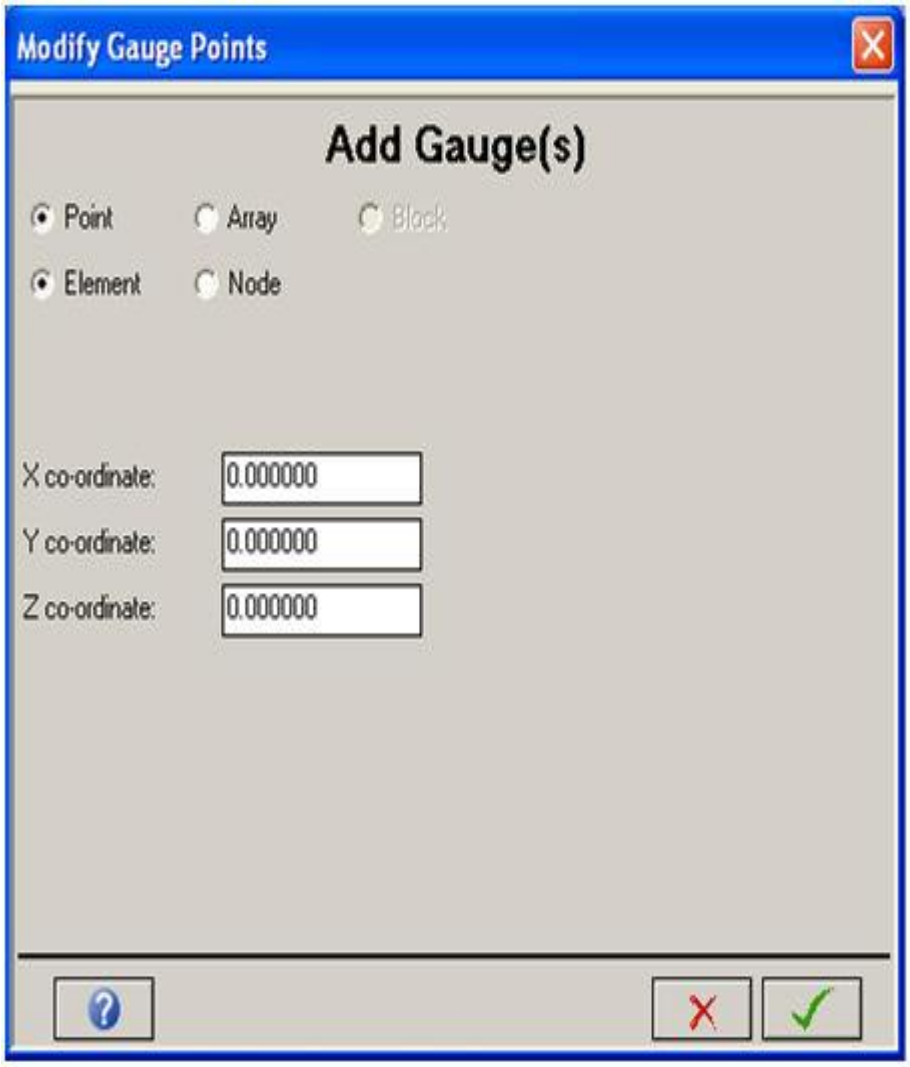


This window lets you delete one or more gauges.

Enter the range of the gauge numbers you want to delete and press

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Gauges - Add Unstructured



This window lets you add gauges to an unstructured part.

A gauge is a location where time histories of selected variables are recorded during a calculation.

Point/Array

Select whether you want to define a single gauge (Point) or a 2D array of gauges (Array)

Element/Node

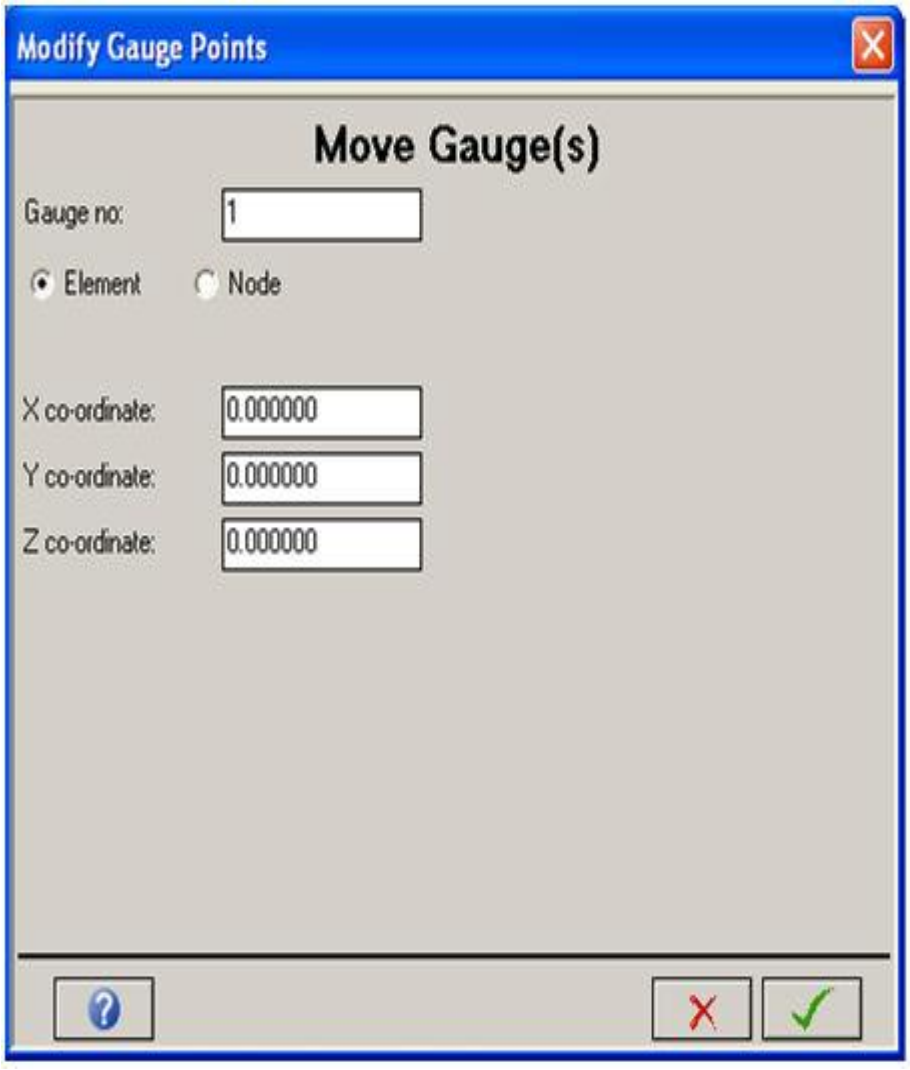
Select whether you want to associate gauge points with element or nodes.

Coordinates

The remaining fields will ask you to specify the (X, Y, Z) coordinates of your gauges.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Gauges - Move Unstructured



This window lets you relocate one gauge point.

Gauge Number

Enter the number of the gauge you want to relocate in this field (gauges are assigned numbers in the order in which you define them).

Element/Node

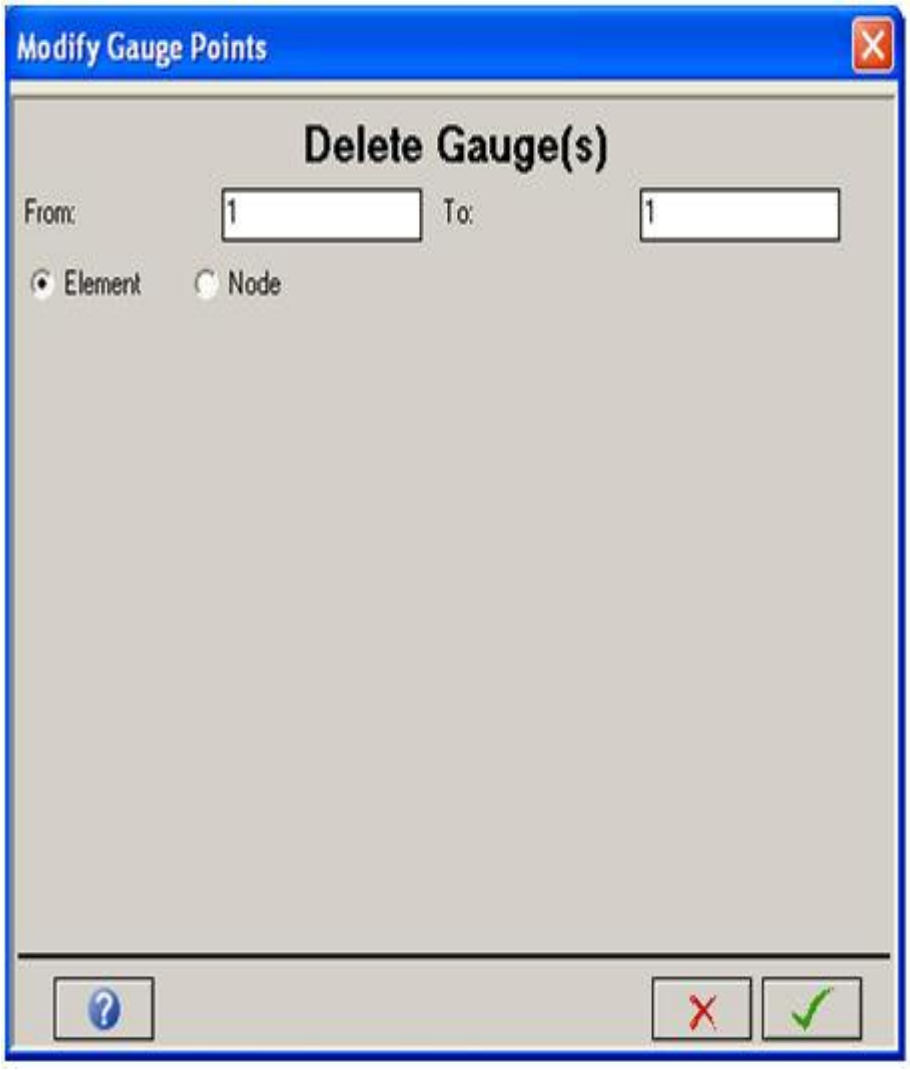
Select whether you want to associate gauge points with element or nodes.

New Position

Enter the (X, Y, Z) coordinates for the new position of the gauge.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Gauges - Delete Unstructured



This window lets you delete one or more gauges.

Enter the range of the gauge numbers you want to delete and press

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Solvers

Parts - Solvers (Structured)

Pressing the **Solvers** button in the **Parts panel** displays the following dialog for structured Parts in the lower region of the panel :

This dialog lets change some solver settings for the current Part.

Type
Press this button to change the type of solver used.

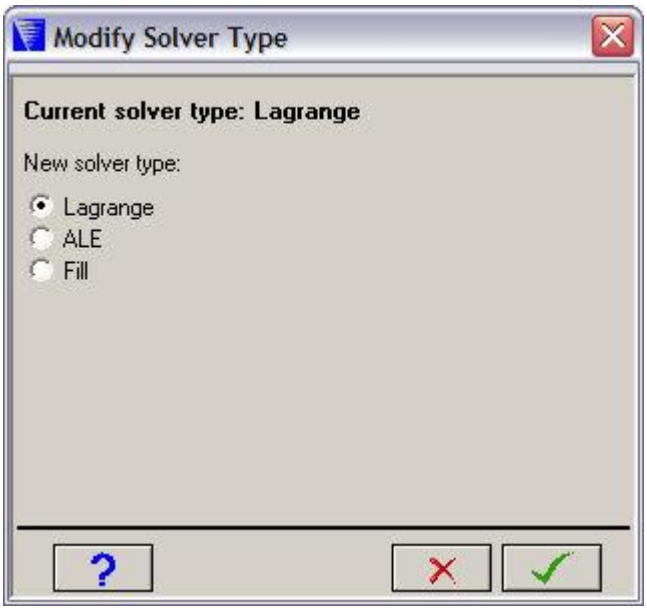
Solution
Press this button to change the solution type associated with some solvers.

Dezone
Press this button to perform dezoning for 2D Lagrange and Euler solvers and for 3D Lagrange, ALE and Euler solvers.

Jetting
Press this button to add the Jetting option to a 2D shell (see the Jetting Tutorial for details).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Type



This window lets you change the type of solver assigned to the current Part.

You can only change to a solver compatible with the current solver used for the Part.

Current solver type

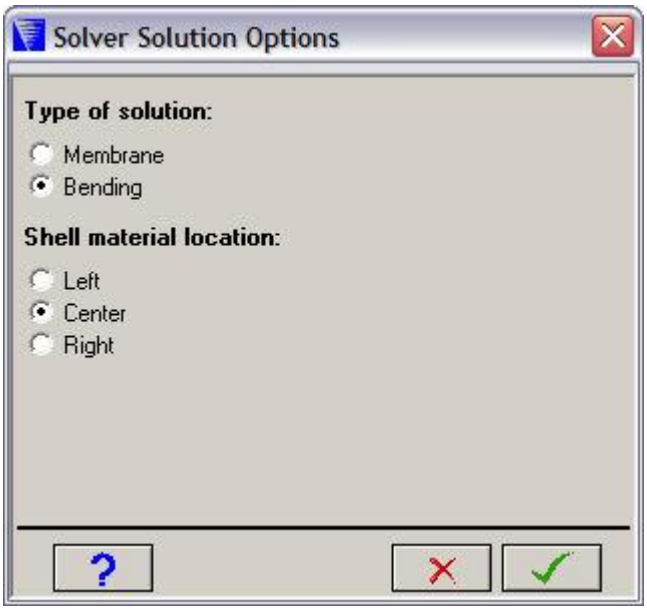
This line shows you the current solver type.

New solver type

A list of the solver types compatible with the current solver type (if any), is shown. Select the solver type you want to use from this list.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Solution (2D Shells)



This window lets you set solution options for 2D shell Parts.

Type of Solution

Select whether you want to compute a **membrane** solution or a solution with full **bending** moments.

Shell material location

Shell segments have no geometrical thickness, so this option lets you select the location of the shell material relative to its geometric position.

Left Material is located to the left of the shell as you progress from the first node to the last node.

Center Material is located equally on either side of the shell (i.e. from -h/2 to h/2, where h is the thickness).

Right Material is located to the right of the shell as you progress from the first node to the last node.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Solution (3D Shells)

This window lets you set solution options for 3D shell Parts.

Type of Solution

Select whether you want to compute a **membrane** solution or a solution with full **bending** moments integrating over 2 or 3 sublayers.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Solutions (Beams)

This window lets you set solution options for Beam Parts.

Type of Solution

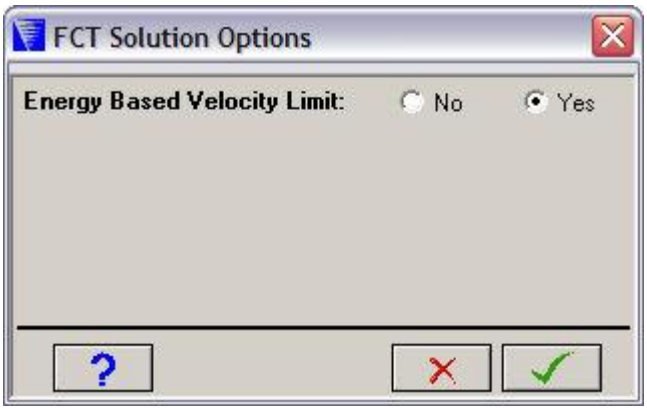
Select whether you want to compute a **truss** solution or a solution with full **bending** moments.

Time-step safety factor

Enter the safety factor you want to apply to the time-step calculation. The default value usually ensures stability of your calculation.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Solution (3D Euler-FCT)



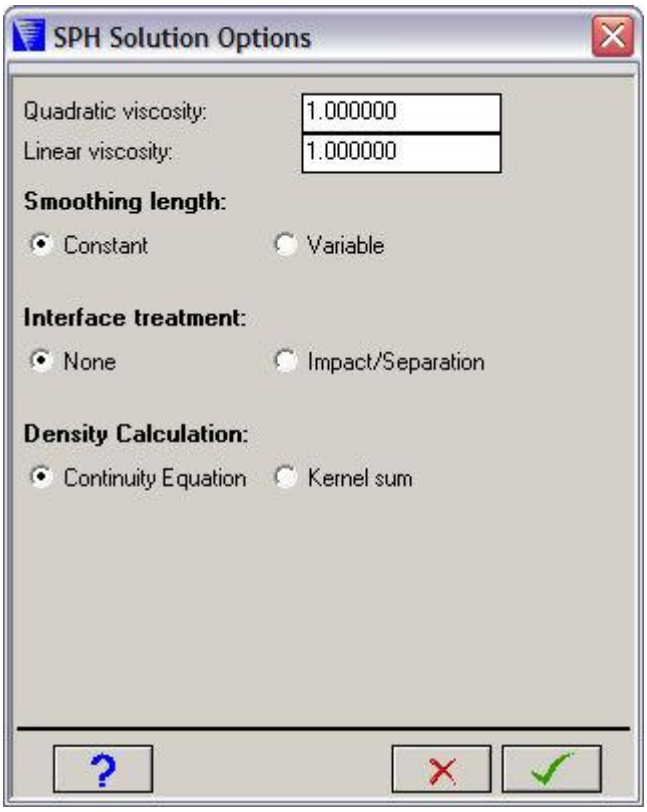
This window lets you set solution options for 3D Euler-FCT Parts.

Energy Based Velocity Limit

Specify whether or not you want to use an energy based velocity limit for Euler-FCT computations.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Solution (2D SPH)



This window lets you set solution options for 2D SPH Parts.

Quadratic viscosity

Enter the coefficient of quadratic viscosity you want to use for SPH computations.

Linear viscosity

Enter the coefficient of linear viscosity you want to use for SPH computations.

Smoothing Length

Specify whether you want to use a **constant** or **variable** smoothing length. Using a variable smoothing length is not recommended for most calculations.

Interface treatment

Specify if you want to use **Impact/Separation** calculations for the treatment of interfaces between different SPH regions.

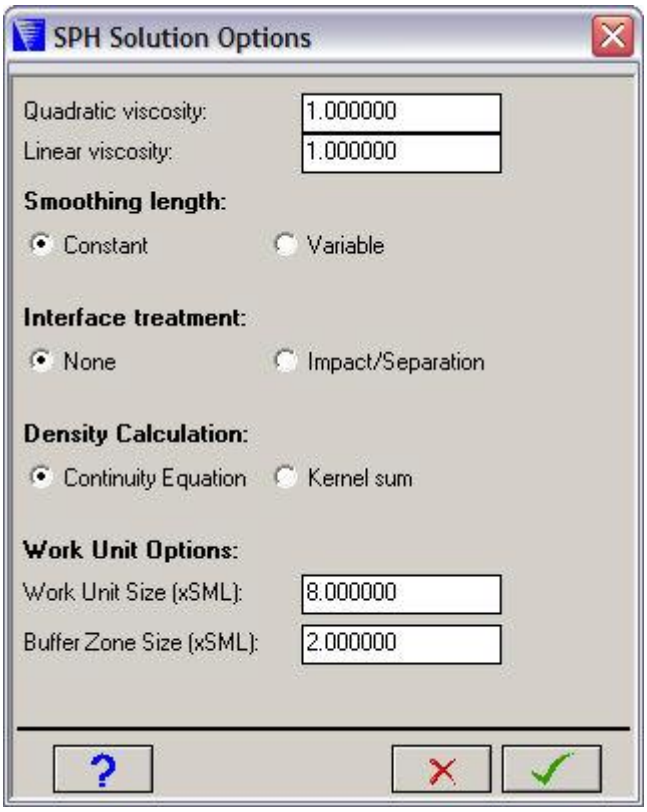
Density Calculation

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Specify whether you want to calculate densities using a **continuity equation** or **kernal sum** method.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Solution (3D SPH)



This window lets you set solution options for 3D SPH Parts. The default settings are the recommended settings.

Quadratic viscosity

Enter the coefficient of quadratic viscosity you want to use for SPH computations.

Linear viscosity

Enter the coefficient of linear viscosity you want to use for SPH computations.

Smoothing Length

Specify whether you want to use a **constant** or **variable** smoothing length. Using a variable smoothing length is not recommended for most calculations.

Interface treatment

Specify if you want to use **Impact/Separation** calculations for the treatment of interfaces between different SPH regions.

Density Calculation

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Specify whether you want to calculate densities using a **continuity equation** or **kernal sum** method.

Work Unit Options

Work Unit Size

Enter the work unit size (as a multiple of the smoothing length) you want to use for SPH calculations. The value you enter here can significantly affect the efficiency of SPH calculations.

Buffer Zone Size

Enter the buffer zone size (as a multiple of the smoothing length) you want to use for SPH calculations. The value you enter here can significantly affect the efficiency of SPH calculations.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Dezone-(2D & 3D Euler)

This window lets you dezone a 2D or 3D Euler (Ideal gas) or Euler (Multi-material) Part.

Dezoning involves combining a number of cells in a fine grid into single cells of a courser grid.

I-factor

Enter the number of cells in the I direction you want to combine into a single cell. This total number of cells in the I direction (IMAX-1) must be exactly divisible by this number.

J-factor

Enter the number of cells in the J direction you want to combine into a single cell. This total number of cells in the J direction (JMAX-1) must be exactly divisible by this number.

Update grid size?

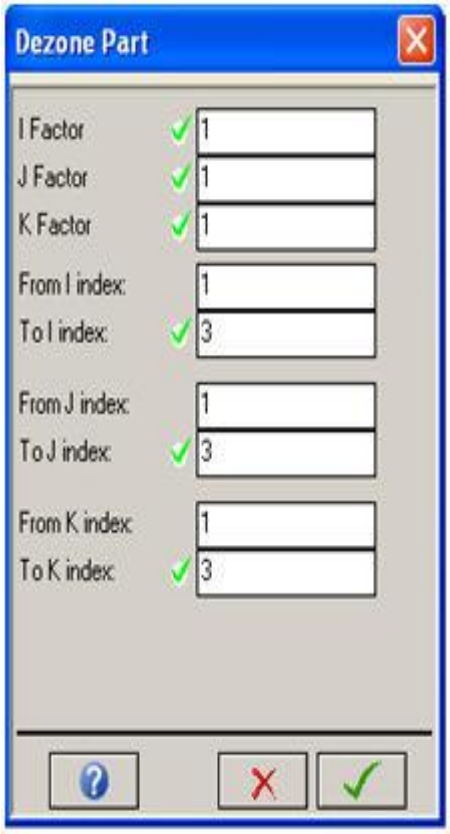
Select **Yes** if you want the IJ index range of the Part to be reduced automatically to reflect the dezone. If you select **No**, the index range will remain unchanged, with the higher indexed cells becoming unused.

K-factor (3D only):

Enter the number of cells in the K direction you want to combine into a single cell. This total number of cells in the K direction (KMAX-1) must be exactly divisible by this number.

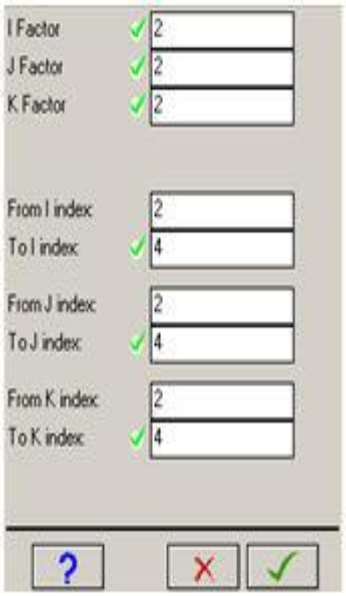
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

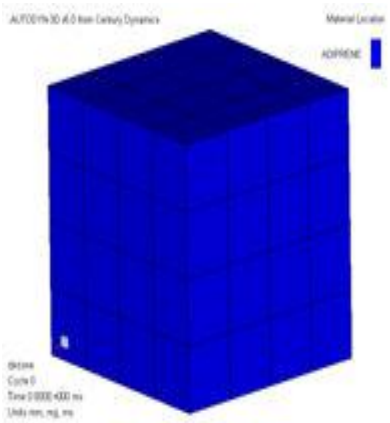
Parts-Solver (Structured) - Dezone (2D & 3D Lagrange/ALE)

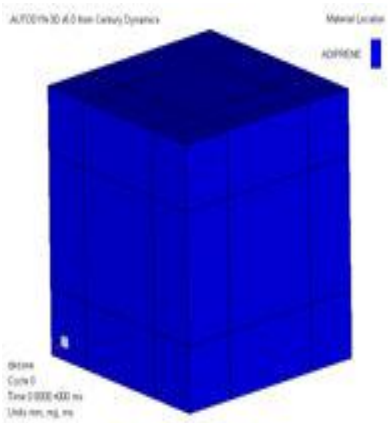


This window lets you dezone a 2D or 3D Lagrange or ALE Part.

Dezoning involves combining a number of cells in a fine grid into single cells of a courser grid.







I-factor

Enter the number of cells in the I direction you want to combine into a single cell. This total number of cells in the I direction (IMAX-1) must be exactly divisible by this number.

J-factor

Enter the number of cells in the J direction you want to combine into a single cell. This total number of cells in the J direction (JMAX-1) must be exactly divisible by this number.

K-factor

Enter the number of cells in the K direction you want to combine into a single cell. This total number of cells in the K direction (KMAX-1) must be exactly divisible by this number.

Dezone Range

In these six fields, enter the range over which the dezone should be performed.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Structured) - Jetting

This window lets you set up jetting options for a 2D Shell Part.

These options let you perform shaped charge jetting analyses in which the liner is modeled as a shell Part and the jet characteristics are computed semi-empirically. The Jetting Tutorial describes these options in detail.

Jetting options:

The top of this window shows the shell nodes you have defined to be jetting nodes.

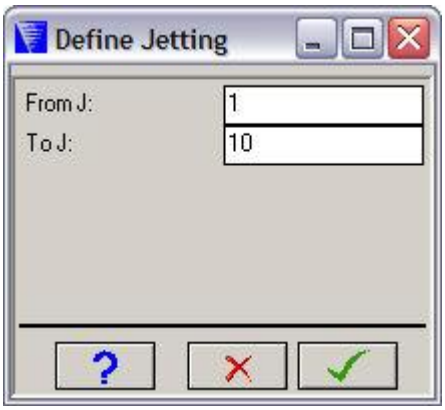
Stop when jetting is done?

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select whether or not you want to stop the calculation once all the jetting nodes have jetted. If you select no, the calculation will continue until the wrapup cycle or time you have set is reached.

Define / Redefine

Press this button to set the shell nodes you want to be jetting nodes. The following window will appear.



Enter the range of shell nodes you want to be jetting nodes.

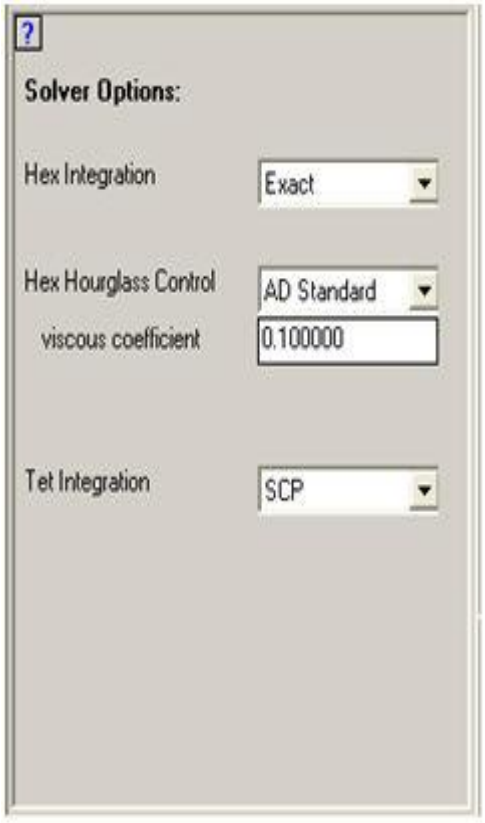
Clear

Press this button to clear (undefine) all current jetting nodes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Unstructured) - Solid

Pressing the **Solvers** button in the **Parts panel** displays the following dialog for unstructured Solid Parts in the lower region of the panel :



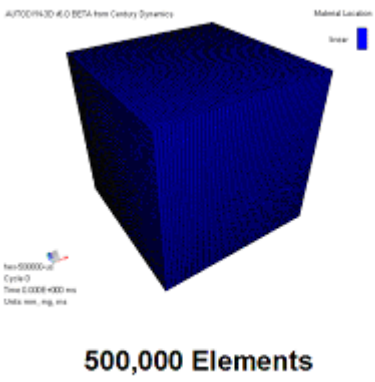
Hex Integration
The unstructured, 8 noded, constant strain, hexahedral element is available in two forms.

- Finite volume formulation with exact volume integration (Wilkins 1974) (Default)
- Finite element formulation using approximate Gaussian quadrature (Hallquist 1976)

The finite volume formulation is the default option and recommended for most applications involving large deformation or warped meshes. This formulation is the same as that used in the Structured (IJK) Lagrange solver.

The finite element formulation is included as a solver option and provides efficiency improvements over the above. However, the accuracy of the element is reduced for warped elements.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



500,000 Elements

	Memory (Mbytes)	Time (Min)	Speed-up
IJK Hex	400	1.74	1.00
New Hex (Exact)	340	0.88	1.97
New Hex (Gauss)	340	0.62	2.80

3.2Ghz Pentium IV Processor, Windows XP

Example performance comparison for the three Hex solvers now available in AUTODYN

Hex Hourglass Control

Hourglass control for the Unstructured Hex solvers is provided in two forms

- AD standard
- Flanagan-Belytschko

The "AD Standard" option is the default option and works well for most applications. This is the most efficient option in terms of memory and speed. A default viscous damping coefficient of 0.1 is recommended.

The "Flanagan-Belytschko" form of hourglass control is available as an option since the AD standard form may not perform well under large rigid body rotations. The Flanagan-Belytschko form of hourglass control is invariant under rotation hence overcomes this problem. Both viscous and stiffness based control is available. The default is stiffness based control with a coefficient of 0.1.

Tet Pressure Interaction

The Unstructured Tet element is available in two forms

- Standard constant pressure (SCP) Tet
- Average nodal pressure (ANP) Tet, (Burton 1996)

The SCP tetrahedral element is a basic, constant strain element and can be used with all the standard AUTODYN material models including erosion. Explosive burn logic is also available. The element is intended as a "filler" element in meshes dominated by hexahedral elements. The element is known to exhibit locking behavior under both bending and constant volumetric straining (i.e. plastic flow). If possible the element should therefore not be used in such cases.

The ANP tetrahedral element is an extension of the advanced tetrahedral element (Burton 1996) and can be used as a majority element in the mesh. The ANP tetrahedral overcomes problems of volumetric locking, which occur with the SCP tetrahedral element. However, the ANP tetrahedral element is still susceptible to shear locking in bending dominated problems. You should therefore be careful to verify their results in such cases.

For meshes containing a majority of tetrahedral elements, the ANP option is recommended.

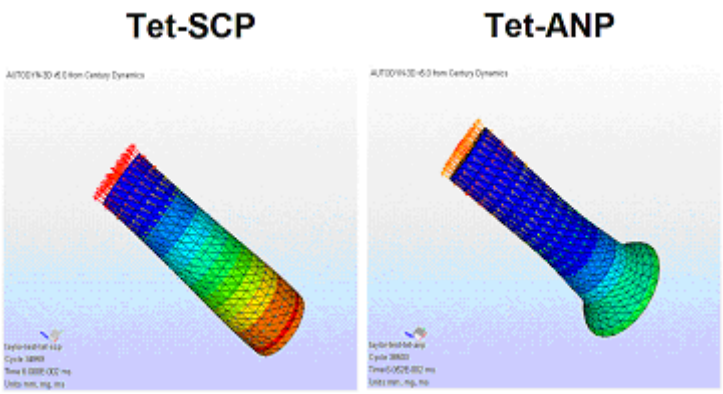
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Material Modeling Options Available with ANP-Tet Elements

Equations of State	Strength Models	Failure Models
Linear	Elastic	Hydro (Pmin)
Polynomial	Viscoelastic	Plastic Strain
Shock	Von Mises	Principal Stress
Porous	Johnson Cook	Principal Stress
Compaction	Piecewise JC	Principal Stress/
P alpha	Zerilli Armstrong	Johnson-Holmquist
Rigid	Steinberg Guinan	RHT Concrete
Hyperelastic	Drucker-Prager	Grady Spall Model
	Johnson-Holmquist	Johnson Cook
	RHT Concrete	Crack Softening
	MO Granular	
	Hyperelastic	

Parts containing ANP tetrahedral elements should only be filled with a single material. Multiple materials can be represented in a single body by creating multiple parts and joining them together.

Note that a hexahedral mesh will generally provide more efficient results than a tetrahedral mesh hence we only recommend the use of predominantly tetrahedral mesh models for convenience of mesh generation.

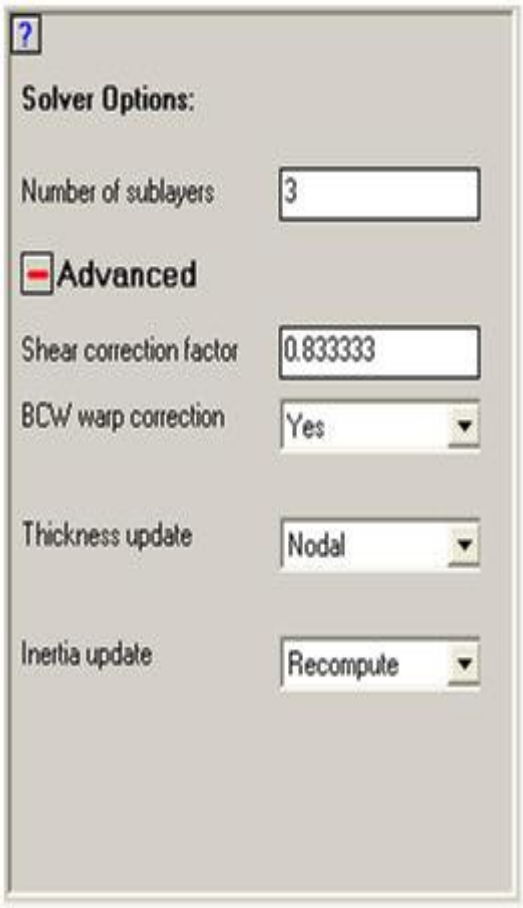


Comparison of SCP and ANP Tet Elements for an impact (Taylor test) involving large amounts of plastic deformation. The ANP-Tet fives results comparable with experiments.

1) Burton A.J., "Explicit, Large Strain, Dynamic Finite Element Analysis with Applications to Human Body Impact Problems", PhD Thesis, University of Wales, December 1996.
2) Bonet J, Burton A.J. "A simple averaged nodal pressure tetrahedral element for incompressible and nearly incompressible dynamic explicit applications". *Communications in Numerical Methods in Engineering 1998;* **14,** 437-449.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Unstructured) - Shell



Number of sublayers
Specify the number of through thickness integration points for an isotropic shell. The default of 3 is suitable for many applications however this number can be increased if better resolution of through thickness plastic deformation/flow is required.

The following advanced options are also available. Note these are not intended for general use.

Shear correction factor
The transverse shear in the element formulation is taken as constant over the thickness. The correction factor accounts for the replacement of the true parabolic variation through the thickness by a uniform transverse shear stress. We do not recommend that you change the default value.

BCW warp correction
The Belytschko-Lin-Tsay element formulation becomes inaccurate if the elements are warped. To overcome this short-coming, the element formulation has a correction for the element warp included. We recommend that this correction is turned on.

Thickness Update

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

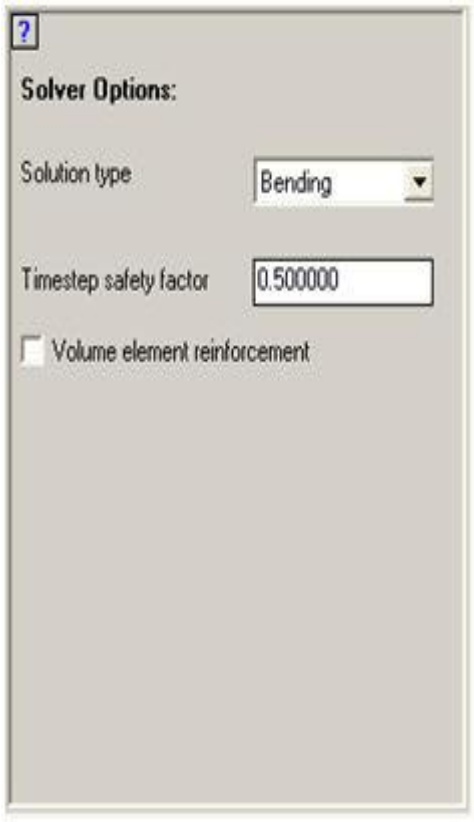
By default the thickness of a shell element is stored and updated at the nodes of the element. An alternative, slightly more efficient, option to store and update the thickness at the element is also provided.

Inertia Update

The principal axes of rotary inertia for shell elements are by default re-calculated each cycle. Alternatively there is an option to rotate the axes, rather than re-compute each cycle. This later option is more efficient however can lead to instabilities, especially in single precision.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Solvers (Unstructured) - Beam



Solution type
The response of a beam element to an applied load includes, by default, the effects of bending moments. This option lets you reduce the beam formulation to that of a truss, in which only axial and torsional effects are considered.

Timestep safety factor
Beam elements require a timestep safety factor that is less than the standard global factor defined under Controls, Timestep. A default value of 0.5 is used and should work for most applications. If problems are encountered with the stability of the beam elements, reducing this factor down to 0.1 is acceptable.

Volume element reinforcement

This option allows reinforcing bars, modeled with beam elements to be tied (bonded) to the volume of a solid element, without the restriction that the nodes of the beams and volume elements initially need to reside at the same physical location. The bonded beam nodes are constrained to stay at the same initial parametric location within the volume element during element deformation. Typical applications involve reinforced concrete or reinforced rubber structures like tires and hoses.

At start-up, AUTODYN will determine the parametric positions of all nodes of beam Parts, selected as reinforcement (Parts, Solver), within volume elements (hex, tet, penta) in the model. The nodes are tied to the parametric co-ordinate of the volume element until the element erodes.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

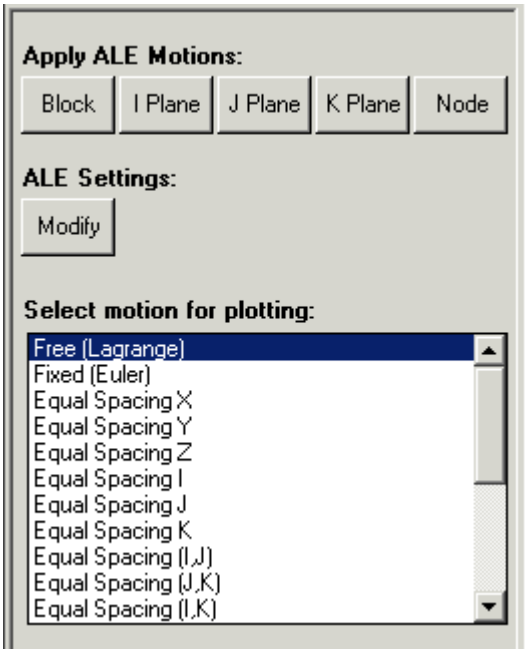
On erosion of the volume element, the beam node bonding constraint is removed and becomes a free beam node.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Motions

Parts - Motions

Pressing the **Motions** button in the **Parts panel** displays the following dialog in the lower region of the panel for structured Parts:



This dialog lets you apply motion constraints to an **ALE** Part.

Block

Press this button to apply motion constraints to an IJK block of nodes.

I/J/K Plane

Press this button to apply motion constraints to an II, J or K plane of nodes.

Node

Press this button to apply a motion constraint to a single node.

Modify (ALE Settings)

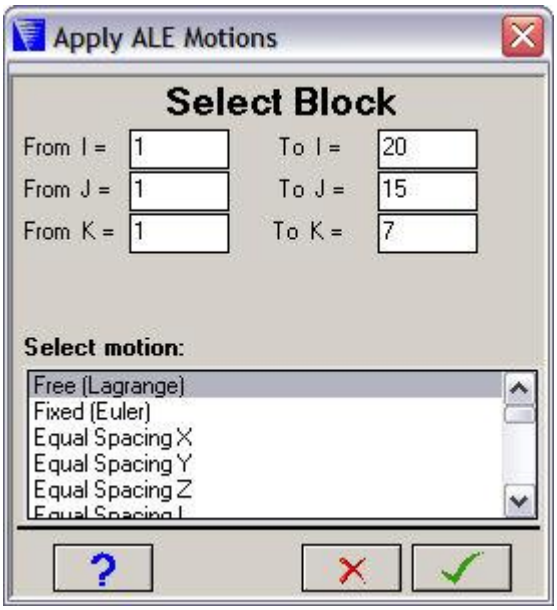
Press this button to change various ALE settings.

Select motion for plotting

Select the motion constraint you want to plot in this scrolling box. All nodes with the selected motion constraint will be highlighted in the plot.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Motions - Block



This window lets you apply an ALE motion constraint to a block of nodes in IJK space.

Select Block

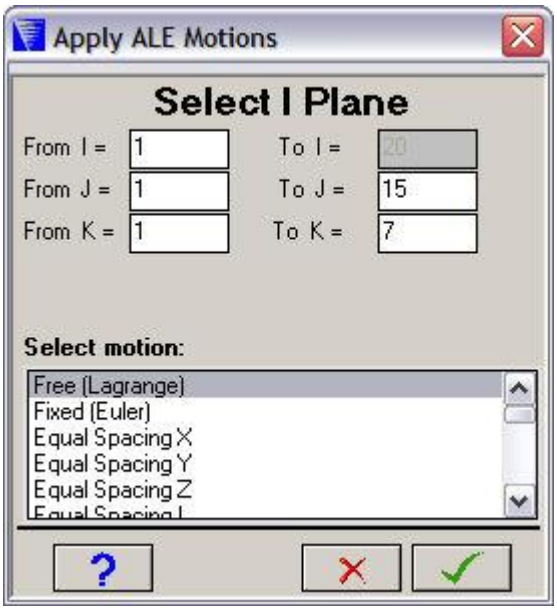
Enter the IJK range for the block to which you want to apply the motion constraint in the fields provided.

Select motion

Select the motion constraint you want to apply from the list provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Motions - I / J / K Plane



This window lets you apply an ALE motion constraint to a plane of nodes in IJK space.
The example shown is for applying a motion constraint to an I plane (input for the other planes is similar).

Select Plane

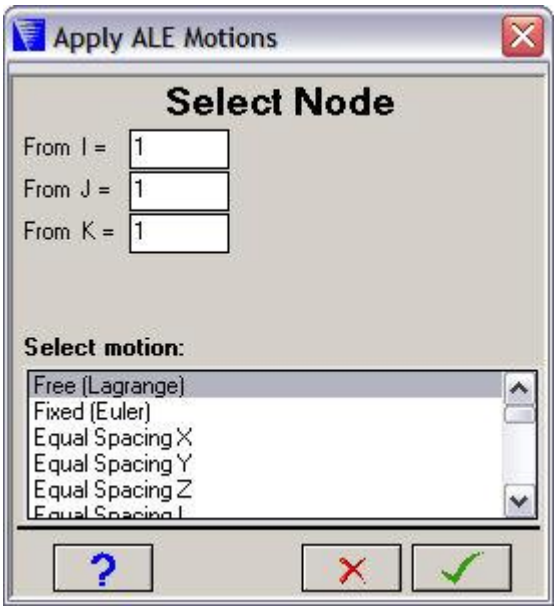
Enter the I index for the plane and the (J,K) range for the nodes to which you want to apply the motion constraint in the fields provided.

Select motion

Select the motion constraint you want to apply from the list provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Motions - Node



This window lets you apply an ALE motion constraint to a node in IJK space.

Select Node

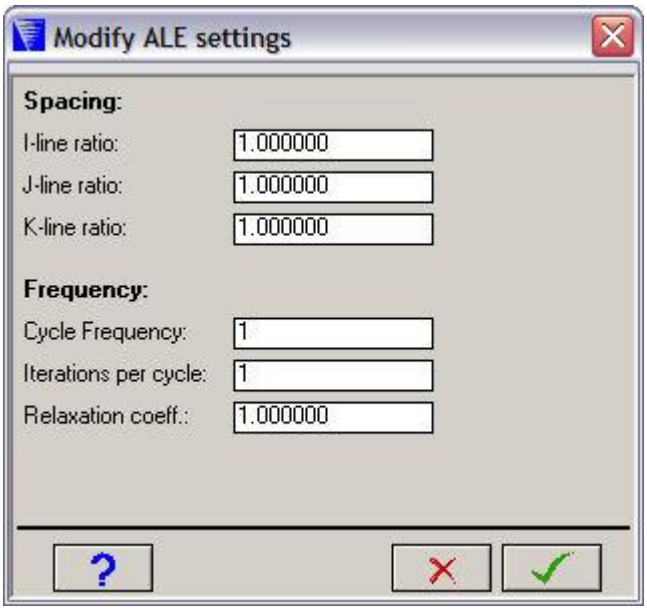
Enter the (I,J,K) indices for the node to which you want to apply the motion constraint in the fields provided.

Select motion

Select the motion constraint you want to apply from the list provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Motions - Modify



This window lets you change ALE settings.

Spacing

Enter the geometric ratios for spacing of I, J and K lines in the fields provided. These are only used for certain motion constraints.

Frequency

Cycle Frequency

Enter how often (in cycles) you want ALE rezoning to be performed.

Iterations per cycle

Enter how many iterations you want the ALE solver to perform in converging to the spacing you have requested. This is used for all motion constraints except **Fixed (Euler)**.

Relaxation coefficient

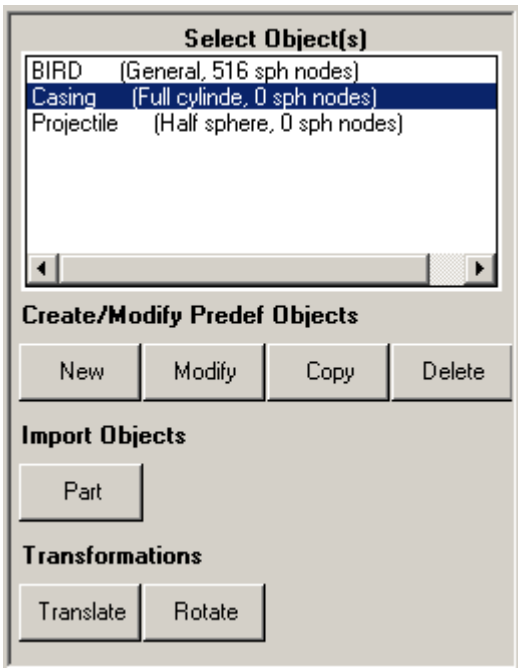
Enter the relaxation coefficient you want to use for the ALE rezoning.
A value of 1.0 will move each node exactly to the location required for its motion constraint.
A value of 0.5 will move each node 50% of the distance towards the location required for its motion constraint.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Geometry

Parts-Geometry - SPH

When SPH parts are selected in the parts menu, pressing the "Geometry (Zoning)" button displays the following panel in the lower region of the parts menu.



This panel allows you to define the geometry for SPH objects.

Select Object(s)

All defined SPH objects are shown in the Select Object(s) list. Select a single object to Modify, Copy or Delete, or select one or more objects to Transform.

Create/Modify Predef Objects

New
Press this button to define a new SPH object.

Modify
Press this button to modify an existing SPH object. This button is only active if a single SPH object is selected. You cannot modify objects created by conversion from a part (using the import objects - part button).

Copy
Press this button to create a copy of the selected SPH object. This button is only active is a single SPH object is selected.

Delete
Press this button to delete the selected SPH object. This button is only active if a single SPH object is selected

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Import Objects

Part

Press this button to covert an existing part (structured or unstructured) into an SPH object. This operation is useful for creating complex SPH objects.

Transformations

Translate

Press this button to translate the currently selected objects.

Rotate

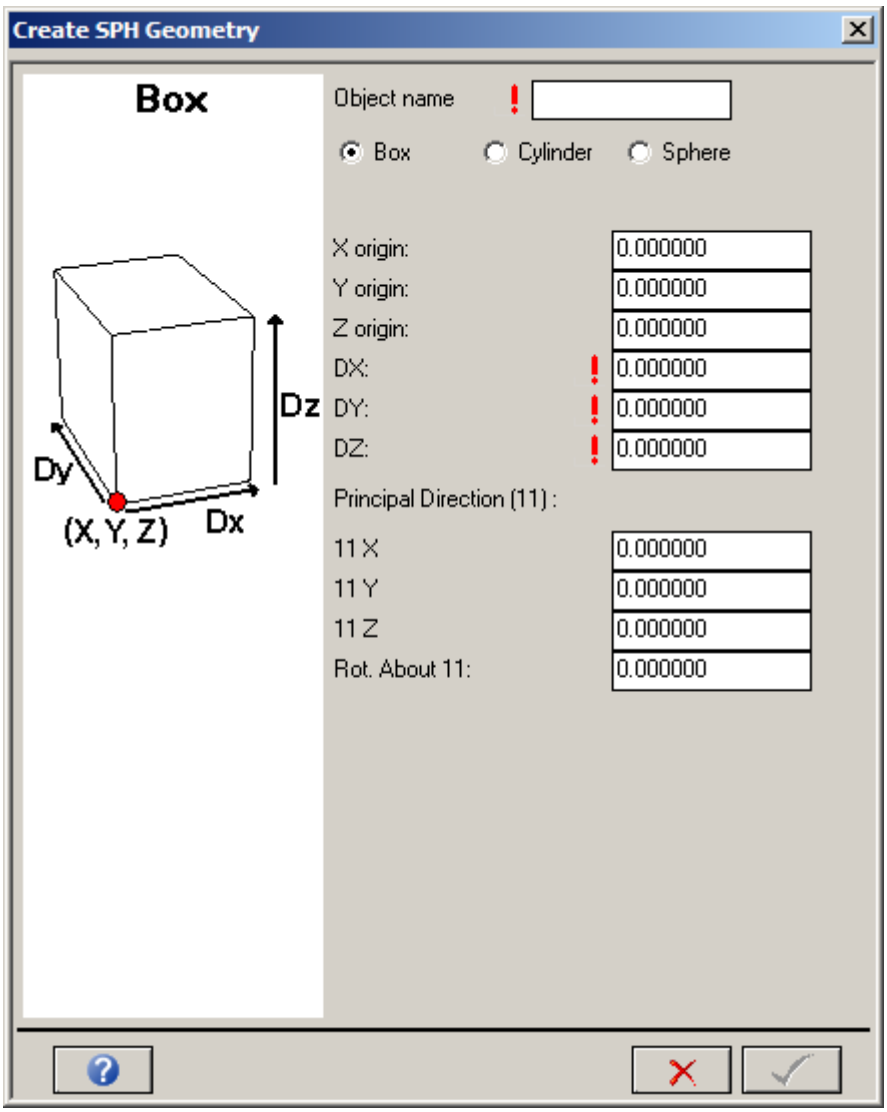
Press this button to rotate the currently selected objects.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts - Geometry - New/Modify

There are three types of Geometry Predef objects you can use - Box, Cylinder and Sphere. These are described below.

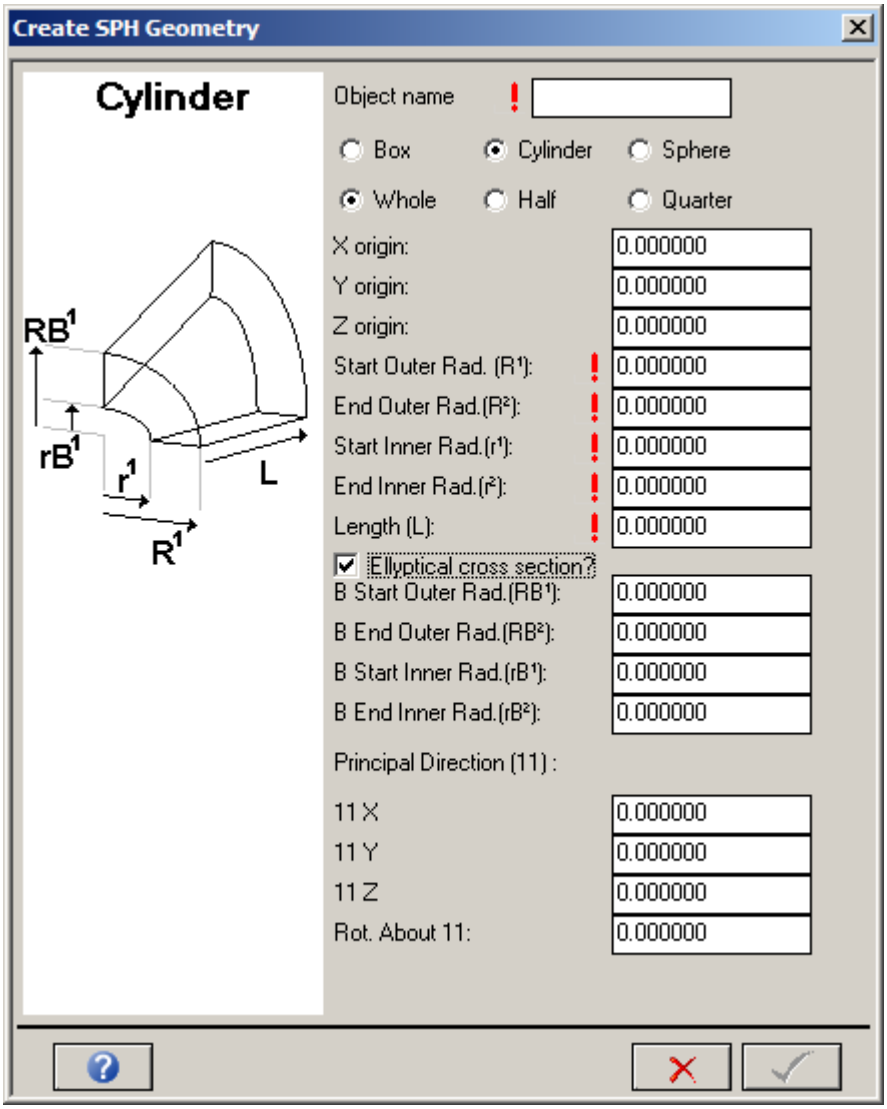
Box



Define the box by entering the XYZ coordinates of the origin, and the size in each of its three directions. The orientation of the box is defined by a principal direction vector and a rotation about this vector. When the box is defined, a wire-frame representation is shown on the screen, together with an arrow displaying the principal vector direction.

Cylinder

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



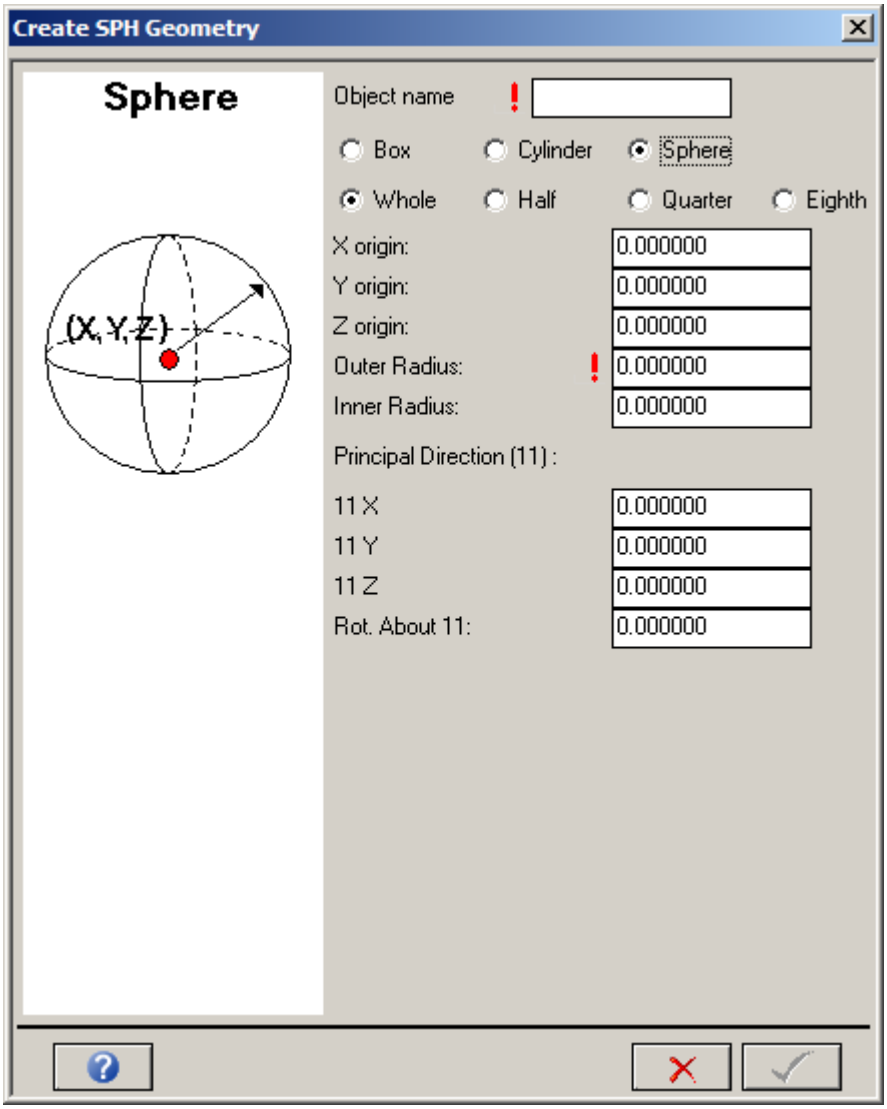
You may define either a whole, half or quarter cylinder by selecting the appropriate option. To define the cylinder, first enter the XYZ coordinates of the start of the cylinder axis. You are then prompted for the start and end radius of the cylinder. If you wish to define a hollow cylinder, enter inner radii for the start and end of the cylinder. Otherwise, to define a solid cylinder leave these two values as zero.

You may also define a cylinder, which is elliptical in cross-section. To do this you must provide extra radii information. This is indicated on the figure on the left hand side of the window.

The orientation of the cylinder is defined by a principal direction vector, which corresponds to the axis of the cylinder and a rotation about this vector. When the cylinder is defined, a wire-frame representation is shown on the screen, together with an arrow displaying the principal vector direction.

Sphere

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

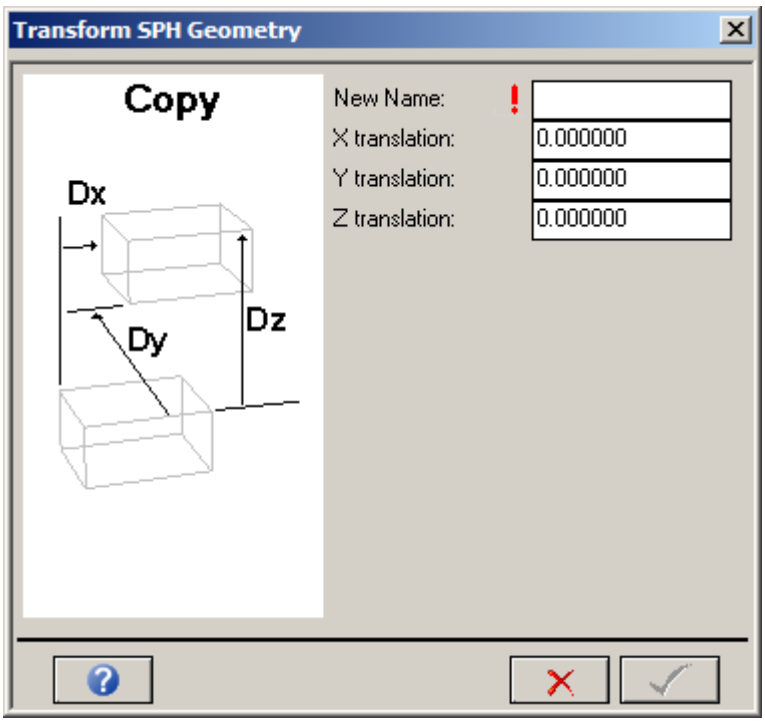


You may define either a whole, half, quarter or eighth sphere by selecting the appropriate option. To define the sphere, enter XYZ coordinates of the sphere origin and an outer radius. You may specify a hollow sphere by entering an inner radius.

The orientation of the sphere is defined by a principal direction vector and a rotation about this vector. When the sphere is defined, a wire-frame representation is shown on the screen, together with an arrow displaying the principal vector direction.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

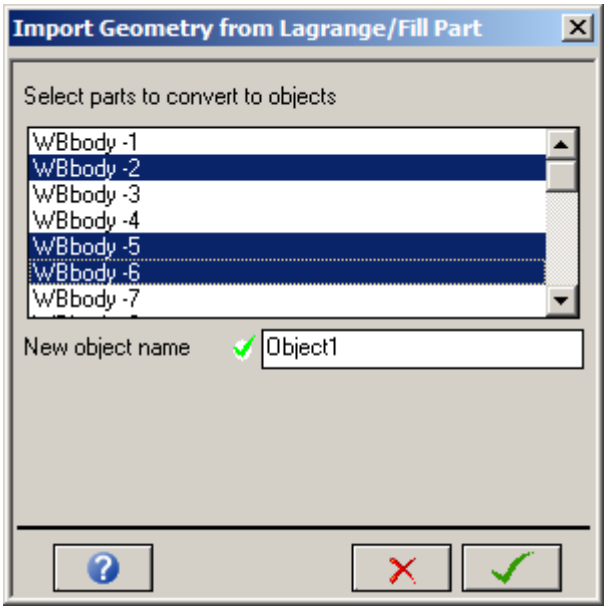
Parts-Geometry-Copy



This window lets you create a copy of the currently selected SPH object. Enter the name of the new SPH object, and also an offset in the X, Y and Z direction.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

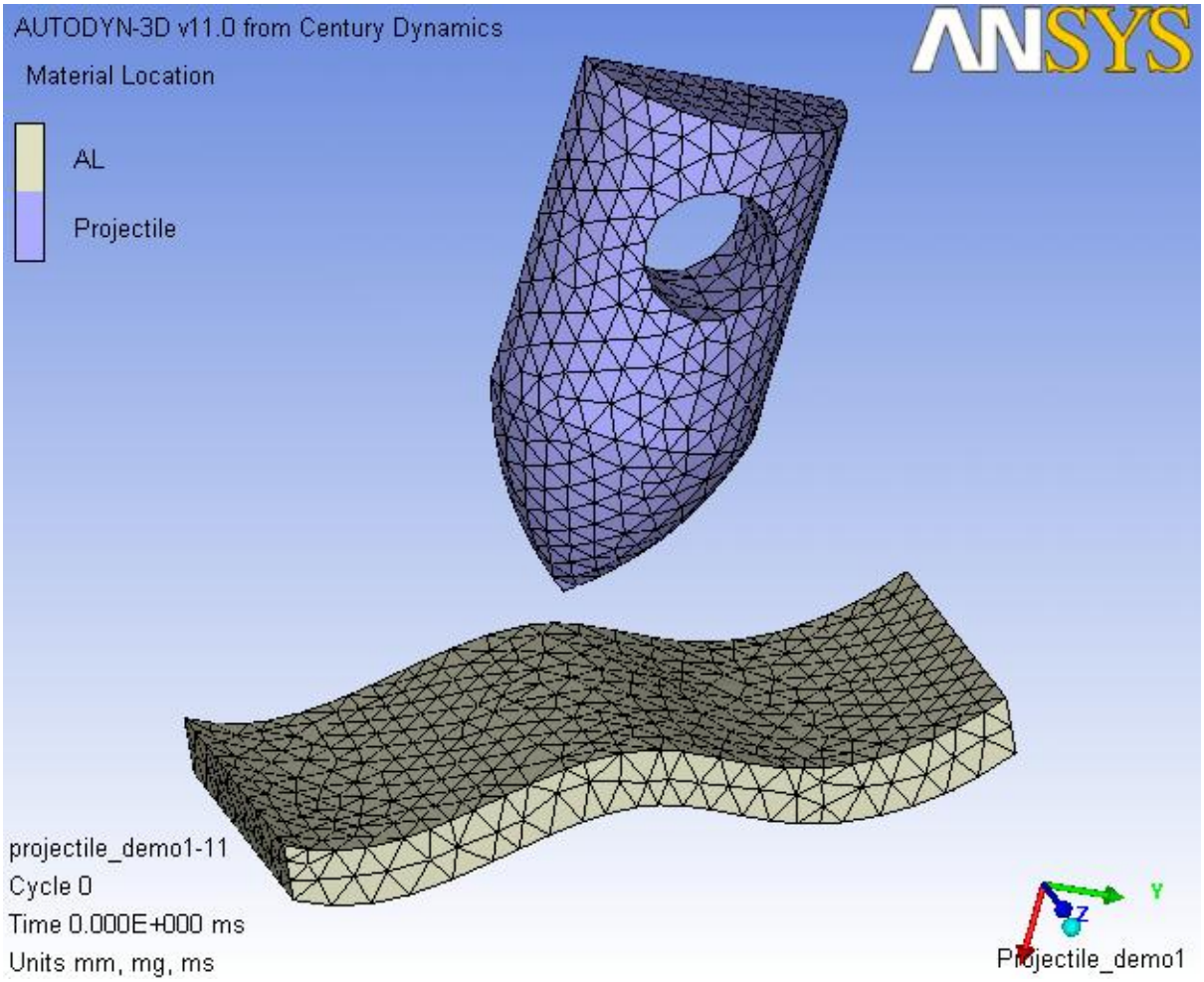
Parts-Geometry-Import Part



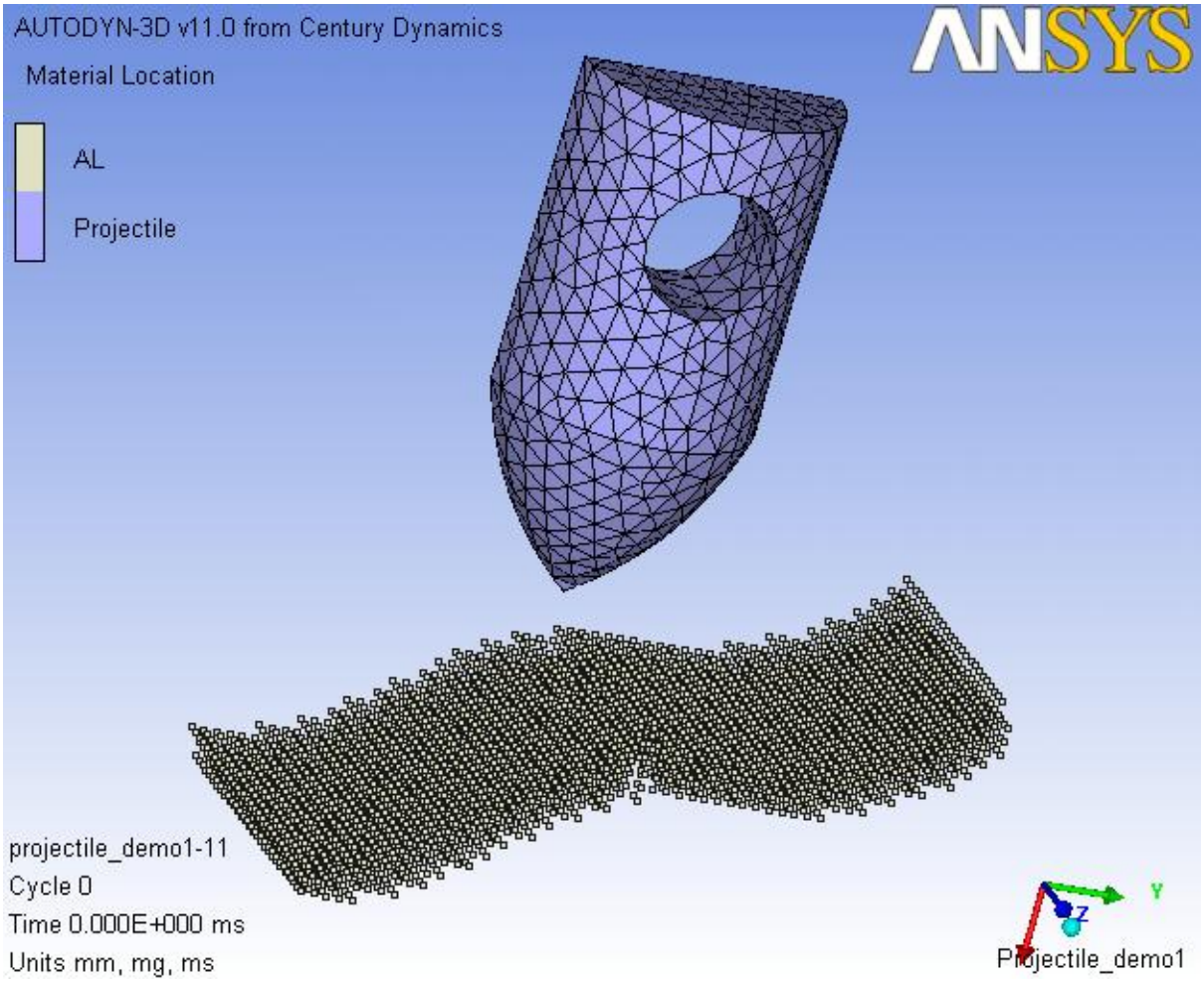
The parts list contains all structured Lagrange or unstructured volume parts in your model. Select one or more of these parts to create an object that can then be packed with SPH particles. You must enter a name for the new object to be created.

The images below show an example of a model with a projectile impacting a target. The target modeled with tetrahedral volume elements in the first image has been converted to an SPH object and packed with SPH nodes. This is shown in the second image.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

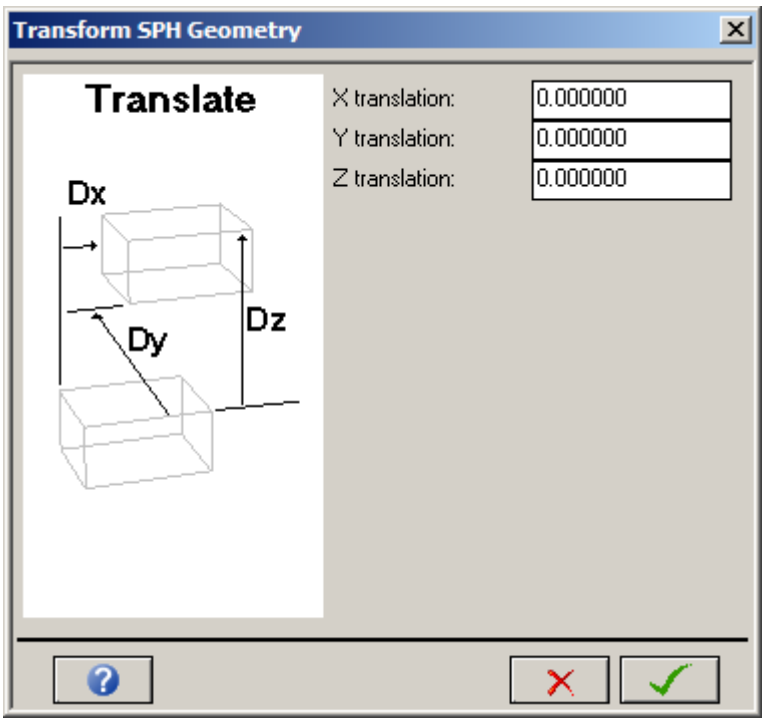


Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

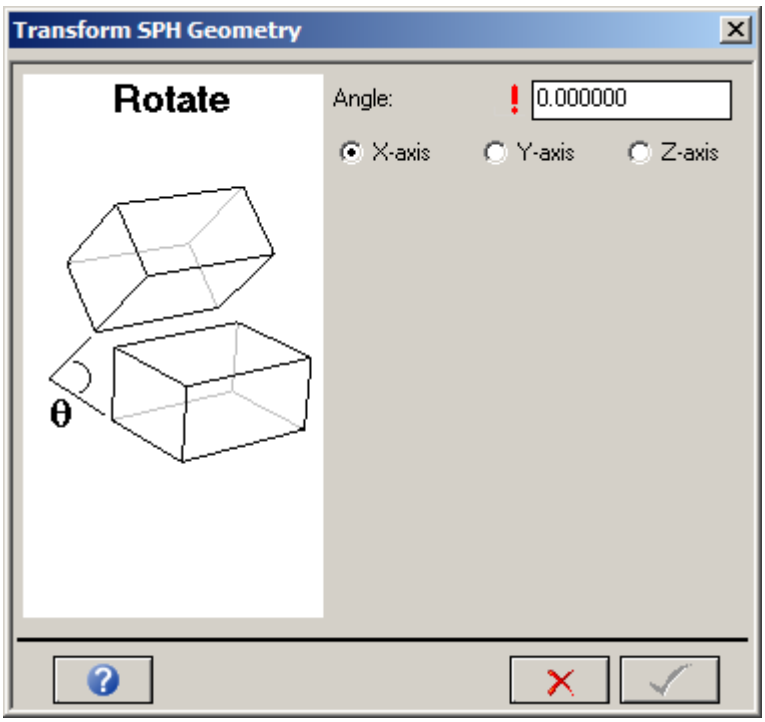
Parts-Geometry-Translate



Enter the distance in the X, Y and Z direction to translate the currently selected part or parts.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parts-Geometry-Rotate

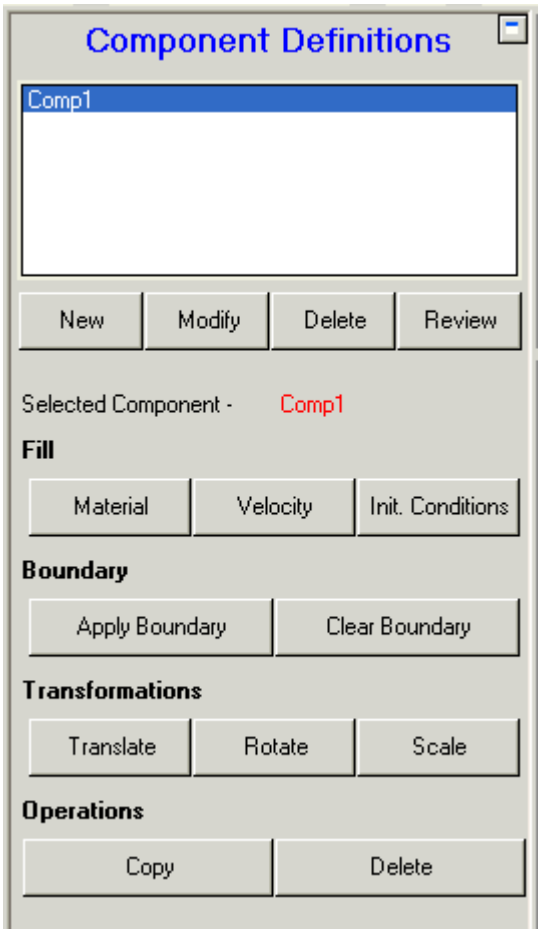


Specify the angle (in degrees) defining the rotation about the selected axis (either X, Y or Z).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components

Components



A Component is a group of structured and unstructured Parts to which you can collectively apply operations.

This panel lets you define Components and use them to perform various operations.

Component List

The top box displays a list of the Components you have defined. You can select a Component in this list.

New
Press this button to create a new Component.

Modify
Press this button to modify an existing Component.

Delete
Press this button to delete a Component.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Review
Press this button to review existing Components.

Fill

Material
Press this button to fill all Parts of the selected Component with a material.

Velocity
Press this button to assign a velocity to all Parts of the selected Component.

Initial Conditions
Press this button to assign initial conditions to all Parts of the selected Component.

Boundary

Apply Boundary
Press this button to apply a boundary condition to the selected Component.

Clear Boundary
Press this button to clear a boundary condition from the selected Component with a material.

Transformations

Translate
Press this button to translate all Parts of the selected Component.

Rotate
Press this button to rotate all Parts of the selected Component.

Scale
Press this button to scale all Parts of the selected Component.

Operations

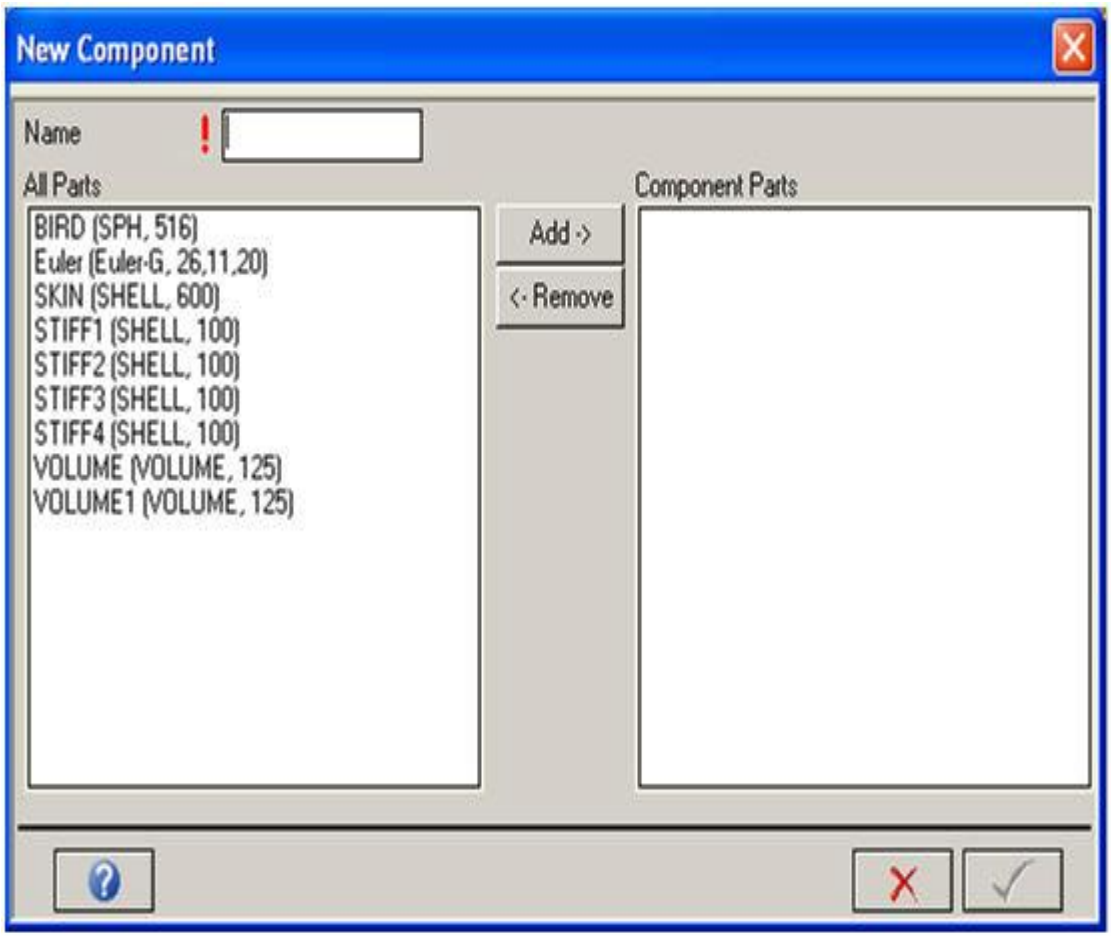
Copy
Press this button to copy the selected Component.

Note that unstructured Parts cannot be copied.

Delete
Press this button delete all Parts of the selected Component.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - New



This window lets you create a new Component.

Name

Enter the name of your Component in this field.

All Parts

This box initially shows a list of all the Parts in your model. You can select one or more Parts in this list.

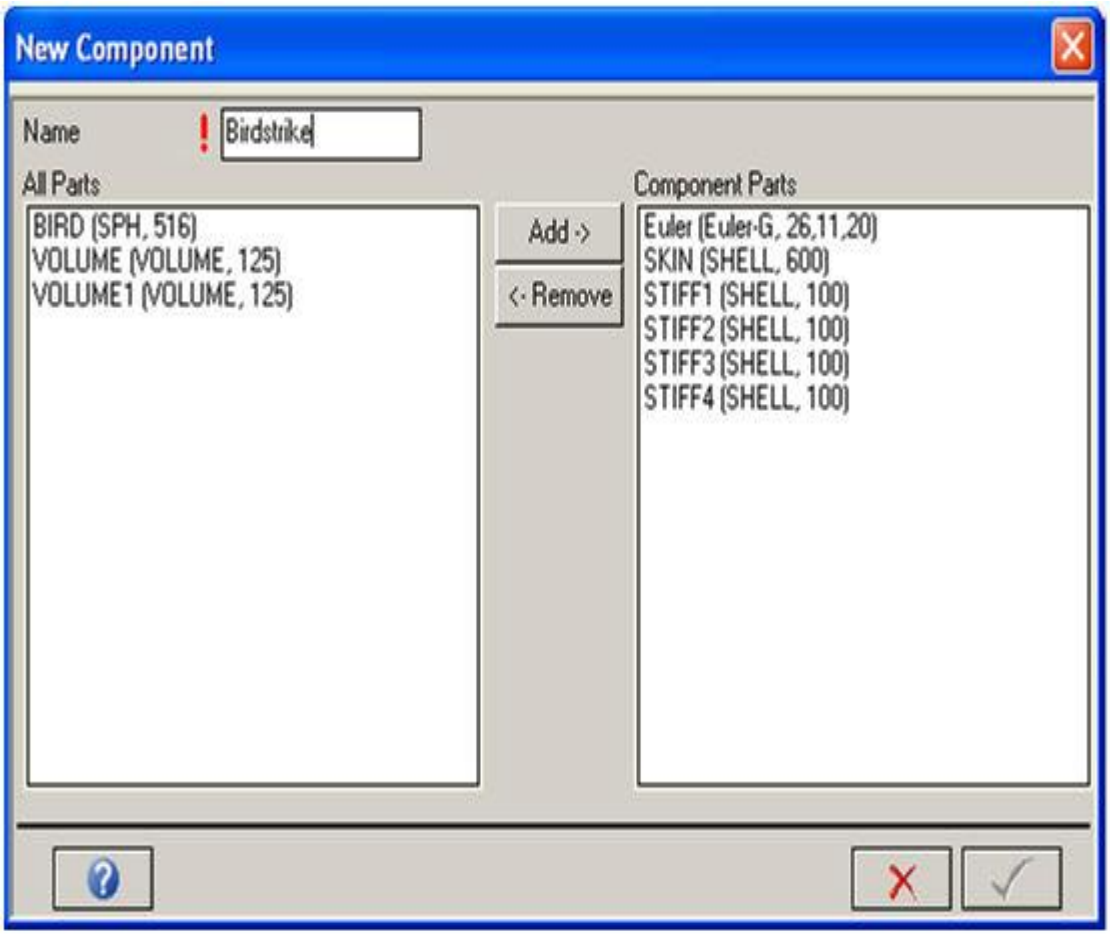
Component Parts

This box (initially empty) shows a list of the Parts you have chosen for your Component.

Add

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select one or more Parts in the left box and press this button to add the selected Parts to your Component.
The Parts will be removed from the list in the left box and added to the list in the right box, as shown in the following example:

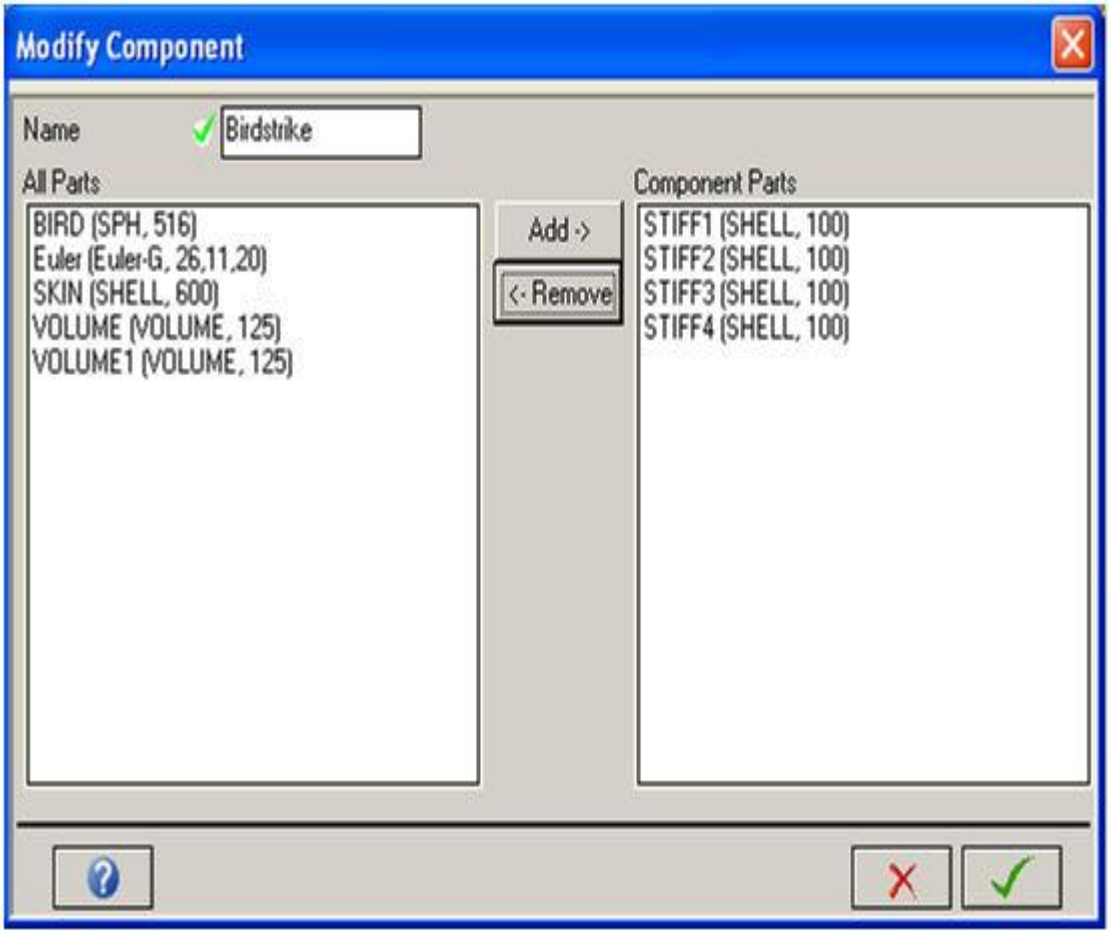


Remove

Select one or more Parts in the right box and press this button to remove the selected Parts to your Component.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Modify



This window lets you modify a Component.

Name

Change the name of your Component in this field.

All Parts

This box shows a list of all the Parts in your model not currently in your Component. You can select one or more Parts in this list.

Component Parts

This box shows a list of the Parts you have chosen for your Component. You can select one or more Parts in this list.

Add

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select one or more Parts in the left box and press this button to add the selected Parts to your Component. The Parts will be removed from the list in the left box and added to the list in the right box.

Remove

Select one or more Parts in the right box and press this button to remove the selected Parts to your Component. The Parts will be removed from the list in the right box and added to the list in the left box.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

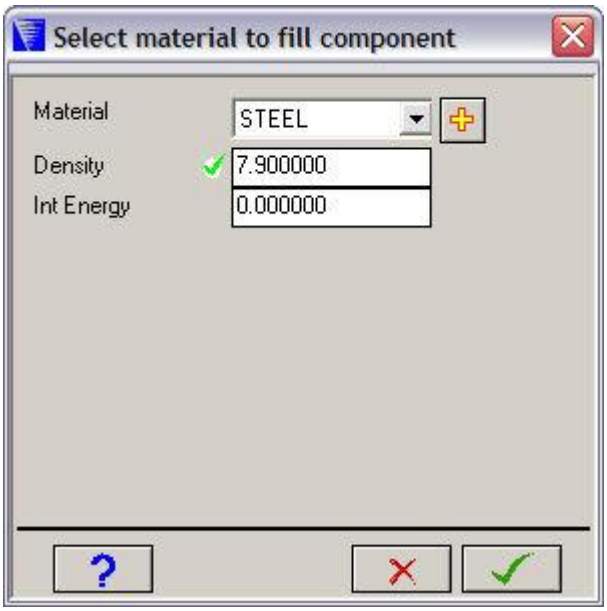
Components - Delete

This window lets you delete one or more components.

Select the components you want to delete in the list and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Material



This window lets you fill all Parts of the selected Component with a material.

Material

Select the material you want to use to fill your Component from this Pull-down menu.

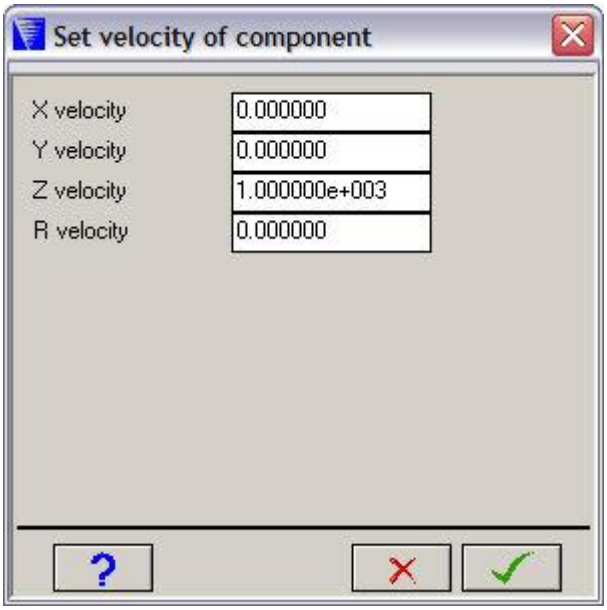
You can define new materials by pressing the **Materials** button on the Navigation Bar or by pressing next to the option in this window.

Material Parameters

Enter material parameter values for your fill in the fields provided (Density, Energy).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Velocity

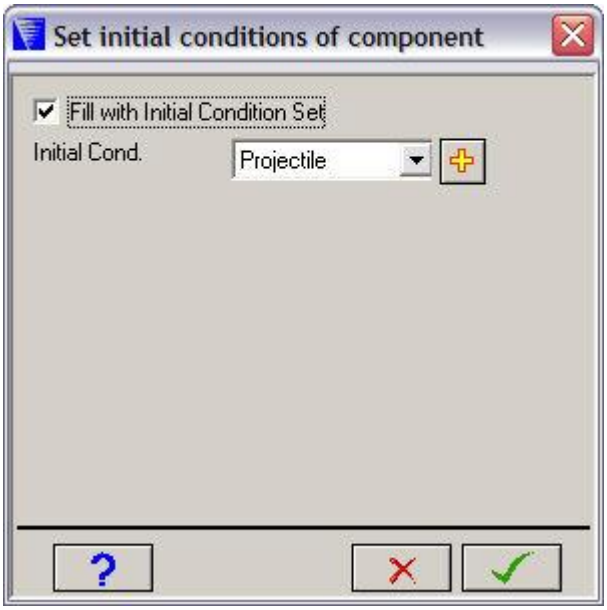


This window lets you assign a velocity to all Parts of the selected Component.

Enter the velocity components in the fields provided and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Initial Conditions



This window lets you assign initial conditions from an Initial Condition Set to all Parts of the selected Component.

Fill with Initial Condition Set

Check this box to fill the Component using an Initial Condition Set.

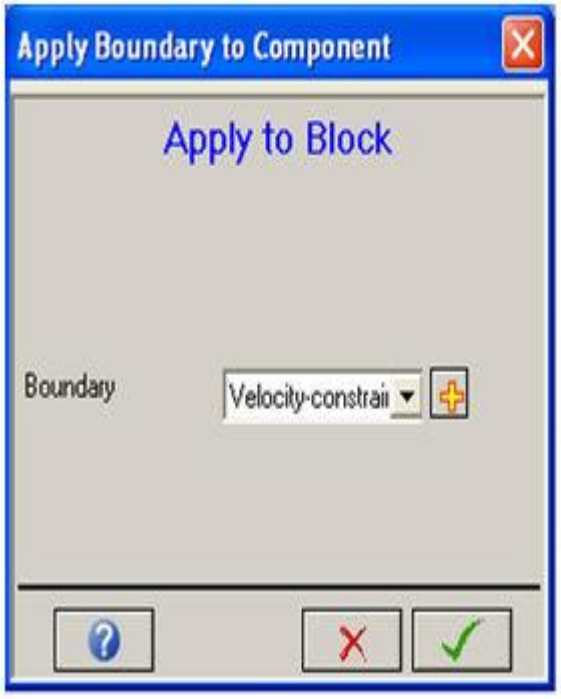
The advantage of filling Parts using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill your Part.

Use the pull-down menu to select the Initial Condition Set you want to use.

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Apply Boundary



This window lets you apply a boundary condition to a Component.

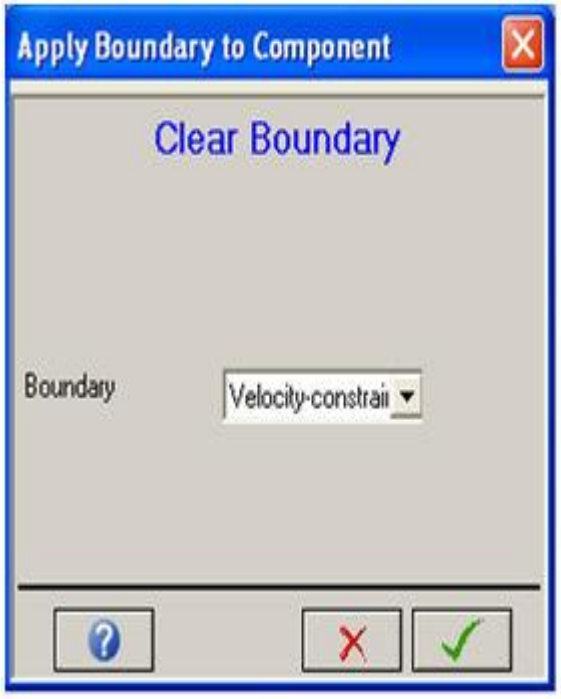
Boundary

Select the boundary condition you want to apply from this pull-down menu.

You can define Boundary Conditions by pressing the **Boundaries** button in the Navigation bar, or you can press next to the pull-down menu.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Clear Boundary



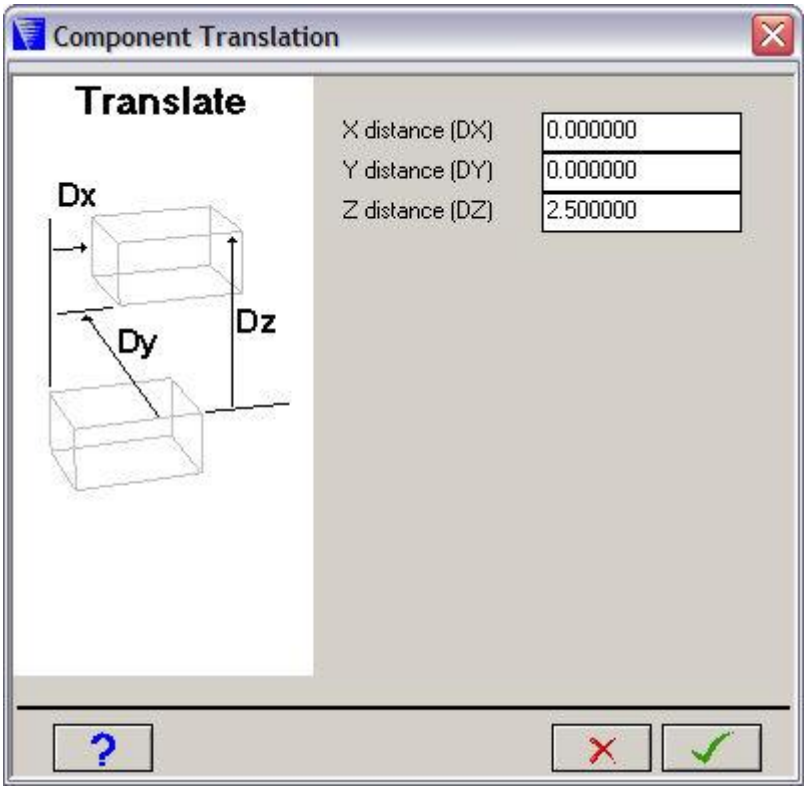
This window lets you clear a Boundary Condition from the current Component.

This operation does not delete the definition of the Boundary Condition, so you can re-apply it later.

Select the boundary condition you want to clear from the pull-down menu and press

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Translate



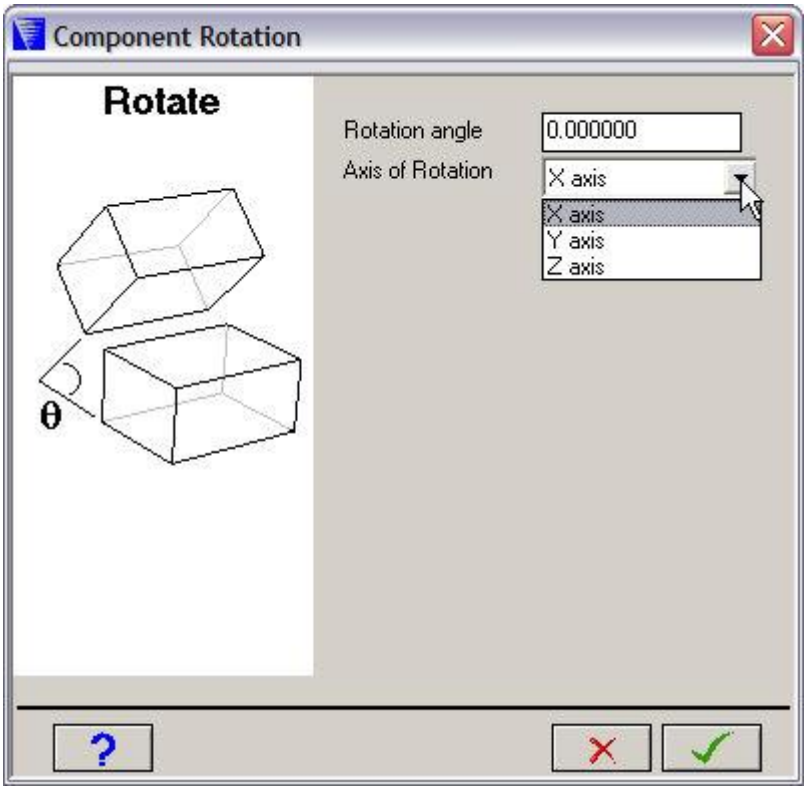
This window lets you translate all Parts of the selected Component in XYZ space.

X/Y/Z distance

Input the X, Y and Z components of the translation in these fields.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Rotate



This window lets you rotate all Parts of the selected Component about an axis in XYZ space.

Rotation

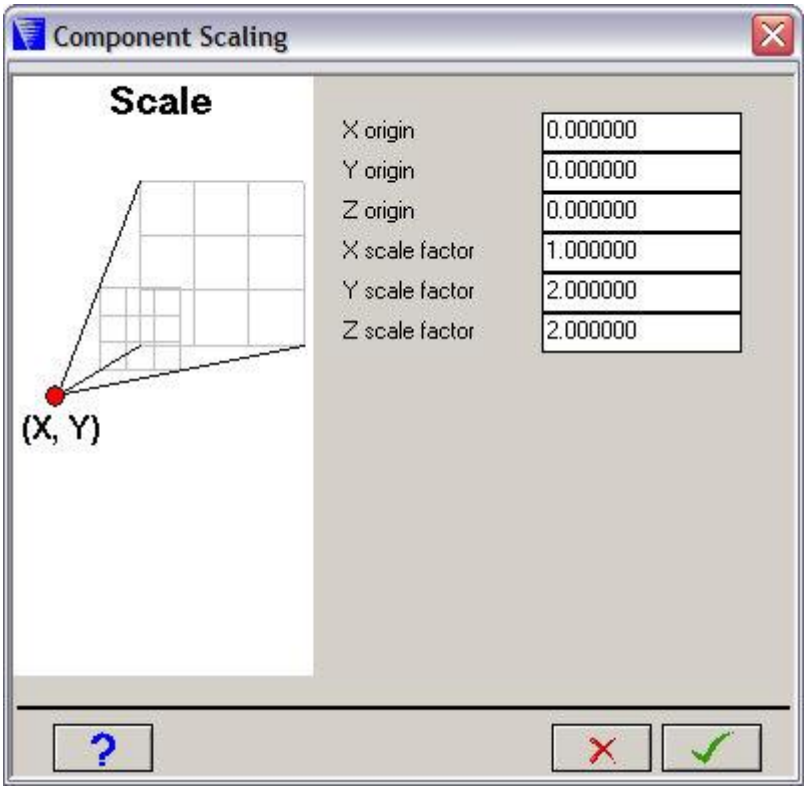
Input the rotation angle in degrees.

X/Y/Z-axis

Use this pull-down menu to specify the axis about which the Component will be rotated.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Scale



This window lets you scale all Parts of the selected Component in XYZ space.

Origin

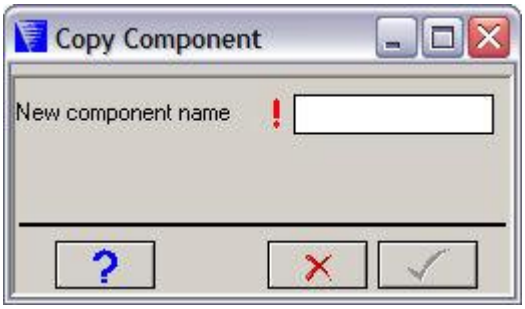
Input the X, Y and Z coordinates of the origin for the scaling.

Scale Factors

Input the X, Y and Z scale factors.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Components - Copy



This window lets you copy the selected Component to a new Component.

During the copy operation a new copy of each structured part in the Component is generated.

Unstructured Parts cannot be copied and therefore will not be present in the copy of the Component.

Enter your name for the new Component and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

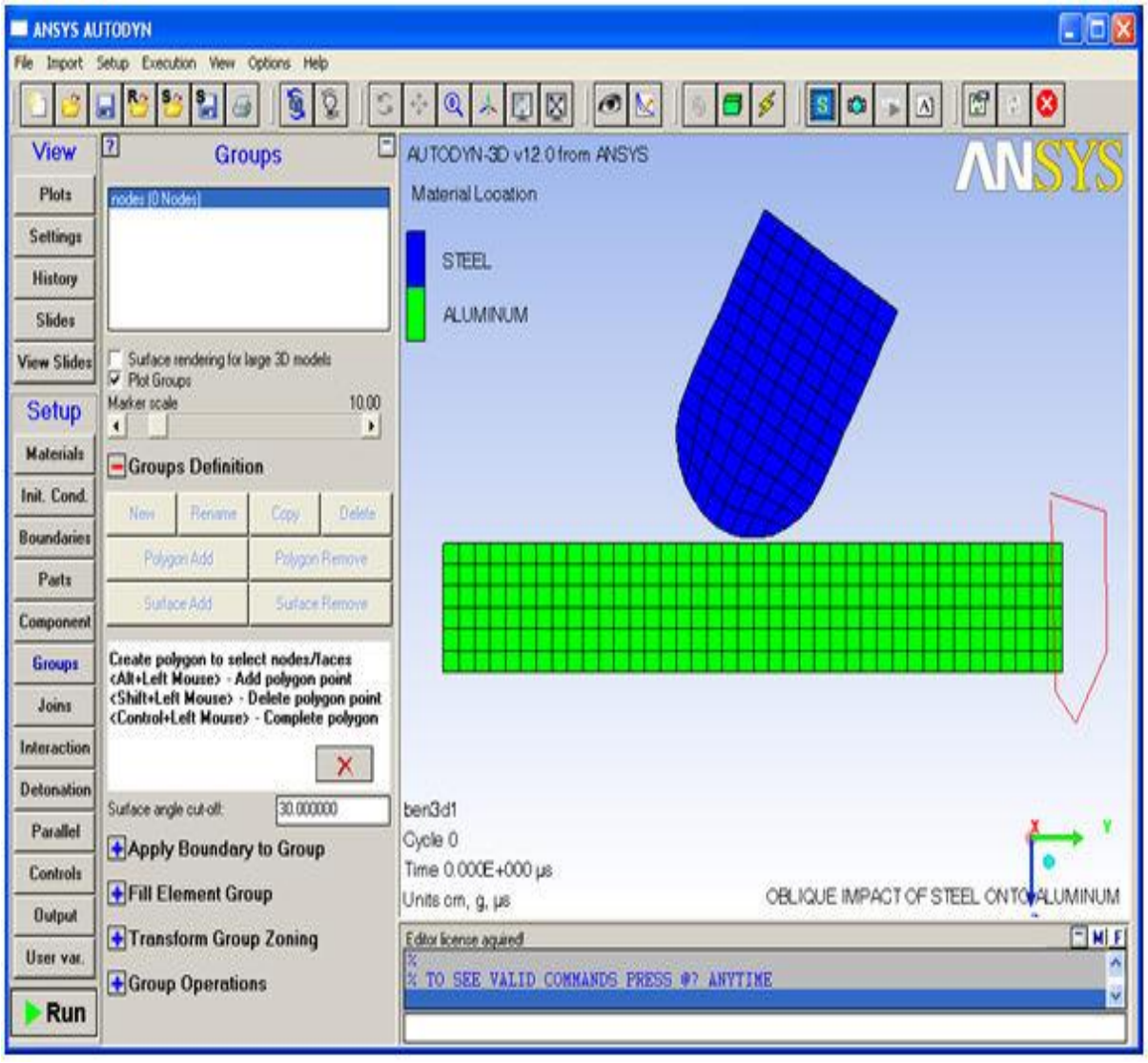
Groups

Groups

Introduction

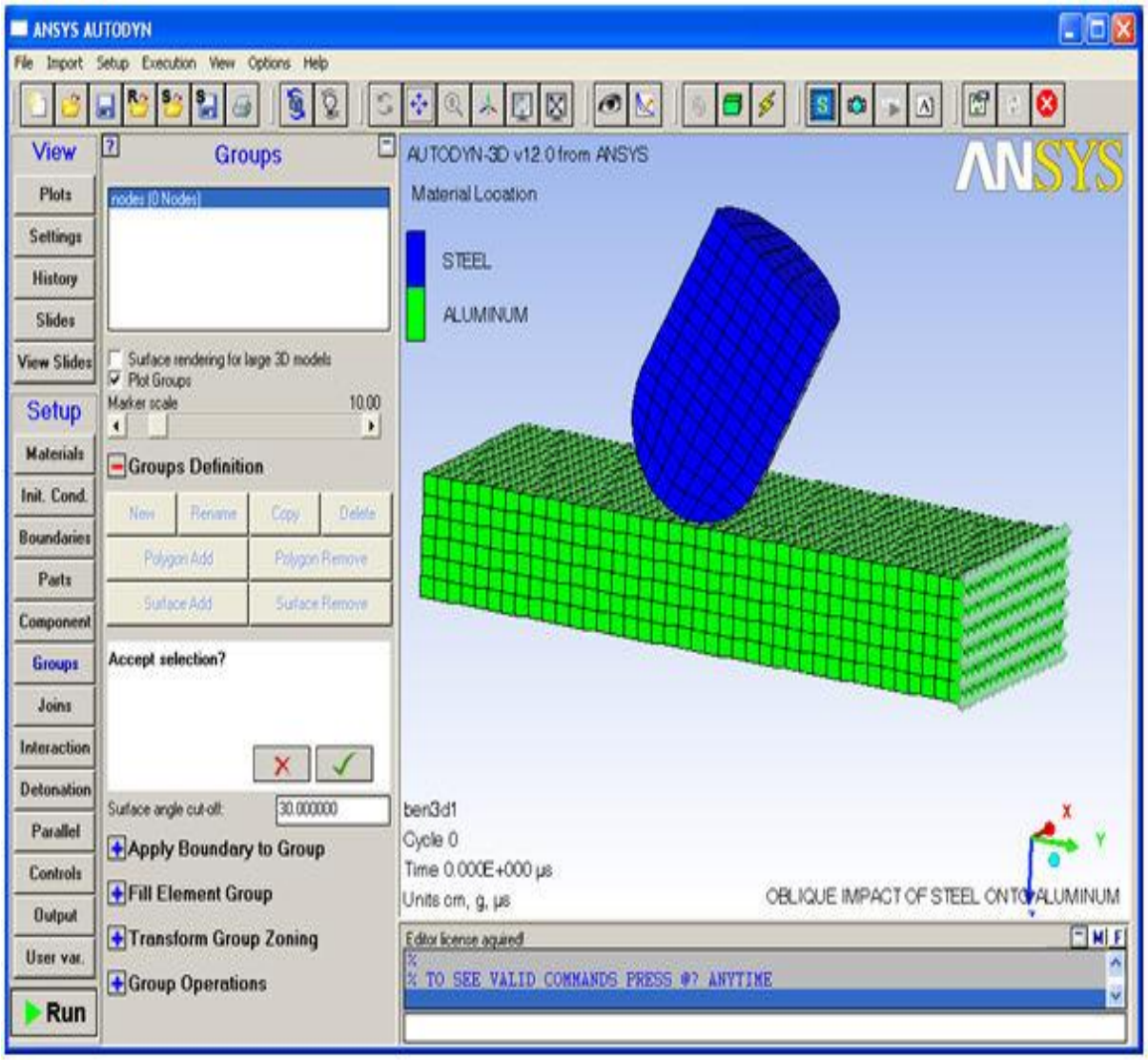
The Group option allows the user to group an arbitrary (independent of Part and Component) selection of nodes faces or elements on which operations can subsequently be performed.

An interactive method of selecting groups of nodes, faces or elements is available. The selection is made by drawing a 2D polygon on the screen around the region of interest, as shown in the figure below.



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

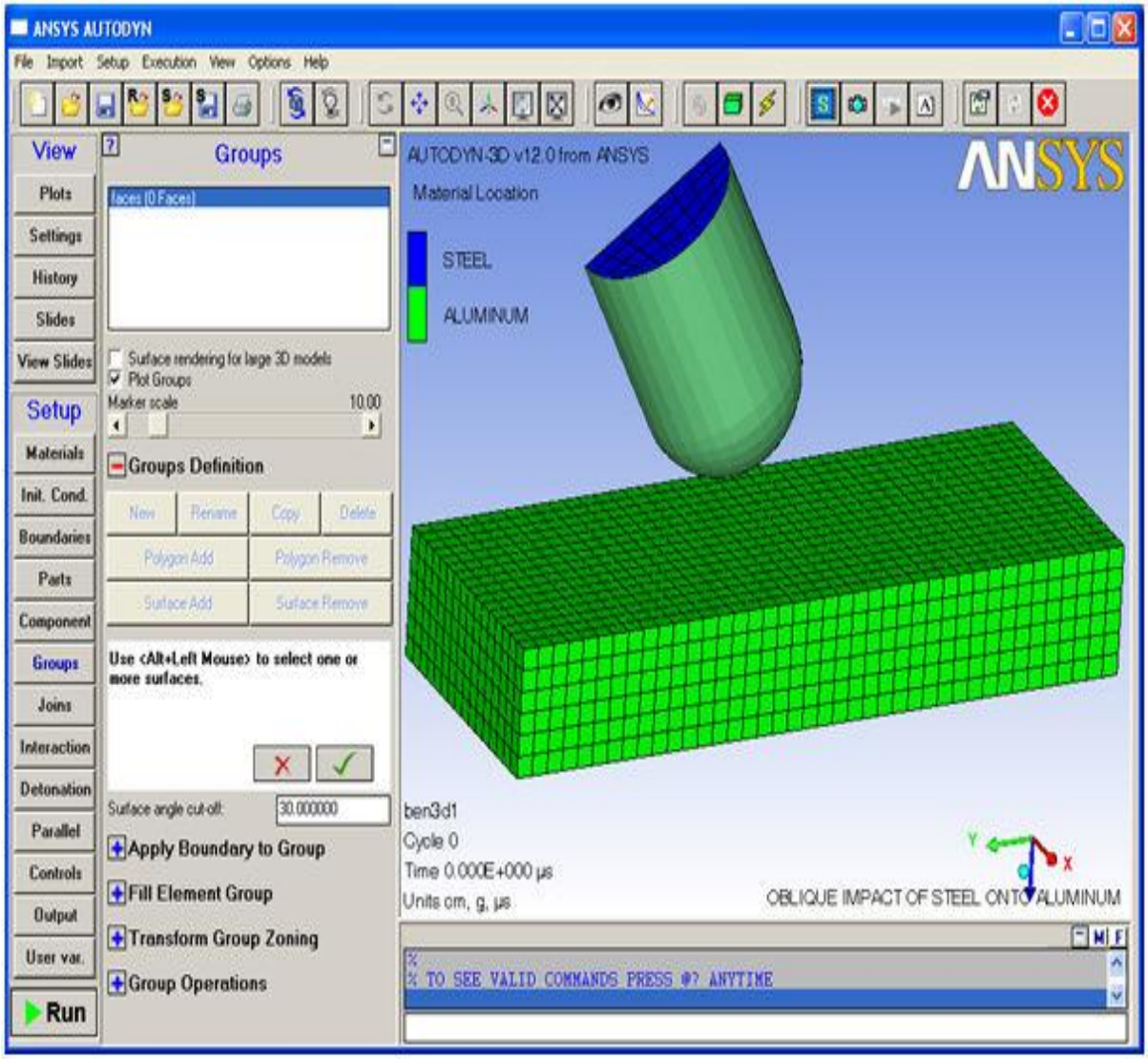
Any nodes, faces or elements that are enclosed by the polygon are highlighted, and the user is prompted to accept the selection of nodes or faces. These are then added to the current group.



For face groups, an alternative surface selection method exists. The selection is made by interactively picking a face element in the graphics window. Any faces that are joined to the selected face and have a normal angle between the two faces of less than a user specified value are then added to the group.

For example, selecting an element on the side of a cube would add all faces on that side of the cube. By setting the surface angle cut-off to a sufficiently large value, curved surfaces may also be selected, as shown in the example below, where the curved surface of the projectile has been added to a group.

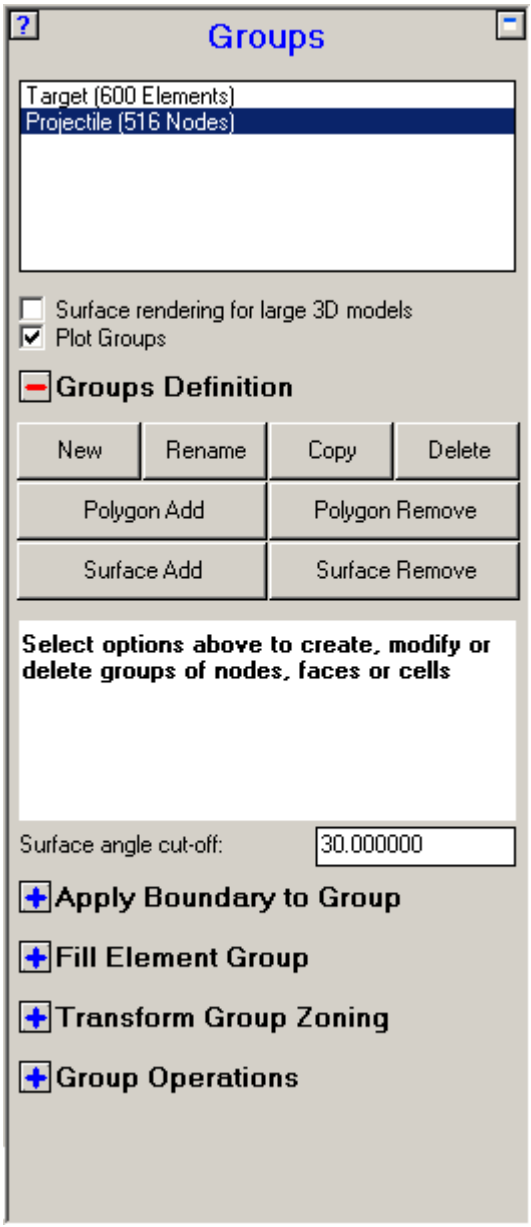
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Once a group is defined, a number of operations can be performed, depending on the type of group.

Groups

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



A Group is a selection of nodes, faces or elements to which you can perform operations, such as applying boundaries, filling.

This panel lets you define Groups and use them to perform various operations.

Group List

The top box displays a list of the Groups you have defined, together with their type (node, face or element) and the group size. You can select a Group in this list.

New

Press this button to create a new Group.

Rename

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press this button to rename an existing Group.

Delete

Press this button to delete a Group

Review

Press this button to review existing Groups.

Groups Definition

Polygon Add
Press this button to interactively define a polygon; any nodes, faces or elements that lie within the 2D polygon are added to the selected Group.

Select the polygon points using the Alt key together with the left mouse button. To delete the last polygon point, use the Shift button together with the left mouse button. To complete the polygon, use the Control and left mouse button.

After completing the polygon, the selected nodes or faces are displayed. Press the button to accept this selection, and add to the current Group. At any stage, the button may be used to cancel the process.

Note that the Polygon Add option is not available (greyed-out) when you have zoomed into the model using the rubber-band option using <Ctrl>+Left mouse button.

To be able to use the Polygon Add option you should use the zoom option, available from the Tool Bar of AUTODYN.

Polygon Remove
Press this button to interactively define a polygon; any nodes, faces or elements that lie within the 2D polygon are removed from the selected Group.

Select the polygon points using the Alt key together with the left mouse button. To delete the last polygon point, use the Shift button together with the left mouse button. To complete the polygon, use the Control and left mouse button.

After completing the polygon, the selected nodes or faces are displayed. Press the button to accept this selection, and add to the current Group. At any stage, the button may be used to cancel the process.

Note that the Polygon Remove option is not available (greyed-out) when you have zoomed into the model using the rubber-band zoom option using <Ctrl>+Left mouse button.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

To be able to use the Polygon Remove option you should use the zoom option, available from the Tool Bar of AUTODYN.

Surface Add
Press this button to choose a surface and add its nodes, faces or elements to the selected Group.

Select a face using Alt and the left mouse button. Any faces that are joined to the selected face and have a normal angle between the two faces of less than a value you specify are then added to the group. For example, selecting an element on the side of a cube would add all faces on that side of the cube. By setting the surface angle cut-off to a sufficiently large value, curved surfaces may also be selected.

After selecting a face, the corresponding surface is highlighted in the graphics window. Press the button to accept this selection or continue adding further surfaces with the Alt and left mouse button. At any stage, the button may be used to cancel the process.

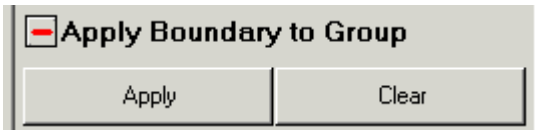
Surface Remove
Press this button to choose a surface and remove its nodes, faces or elements to the selected Group.

Select a face using Alt and the left mouse button. Any faces that are joined to the selected face and have a normal angle between the two faces of less than a value you specify are then removed from the group. For example, selecting an element on the side of a cube would remove all faces on that side of the cube. By setting the surface angle cut-off to a sufficiently large value, curved surfaces may also be selected.

After selecting a face, the corresponding surface is highlighted in the graphics window. Press the button to accept this selection, or continue removing further surfaces with the Alt and left mouse button. At any stage, the button may be used to cancel the process.

Surface Angle Cut-Off
Enter a cut-off angle (in degrees). Only faces with an angle between the normal and a neighbor face's normal of less than this value will be included in a surface selection.

Apply Boundary to Group



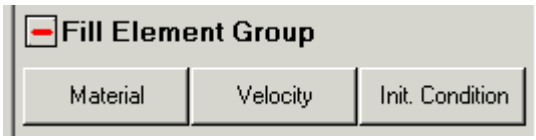
Apply

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press this button to apply a boundary condition to a node or face Group.

Clear
Press this button to remove a boundary condition from a node or face Group.

Fill Element Group



Material
Press this button to fill the selected element Group with a material.

Velocity
Press this button to assign a velocity to the selected element Group.

Initial Conditions
Press this button to assign initial conditions to the selected element Groups.

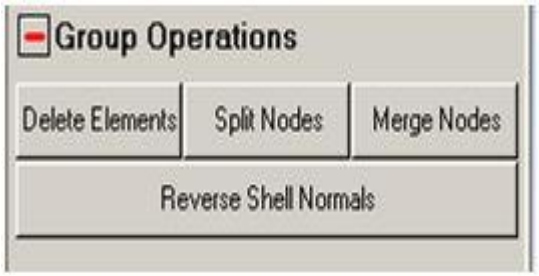
Transform group zoning

Translate
Press this button to rotate the current group

Rotate
Press this button to rotate the current group

Scale
Press this button to scale the current group

Group Operations

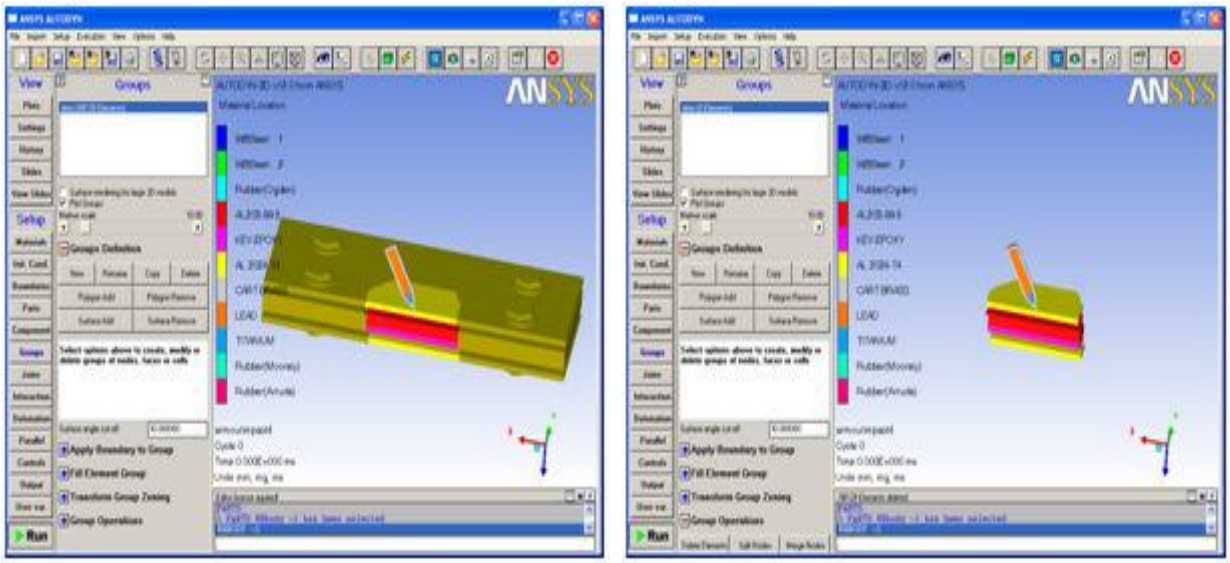


Four operations are available for element groups.

Delete Elements

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Use this option to delete all elements within a selected Group. Select the Group containing the elements to delete and select OK. The elements will be deleted from the model.



Define Element Group Delete Elements in Group

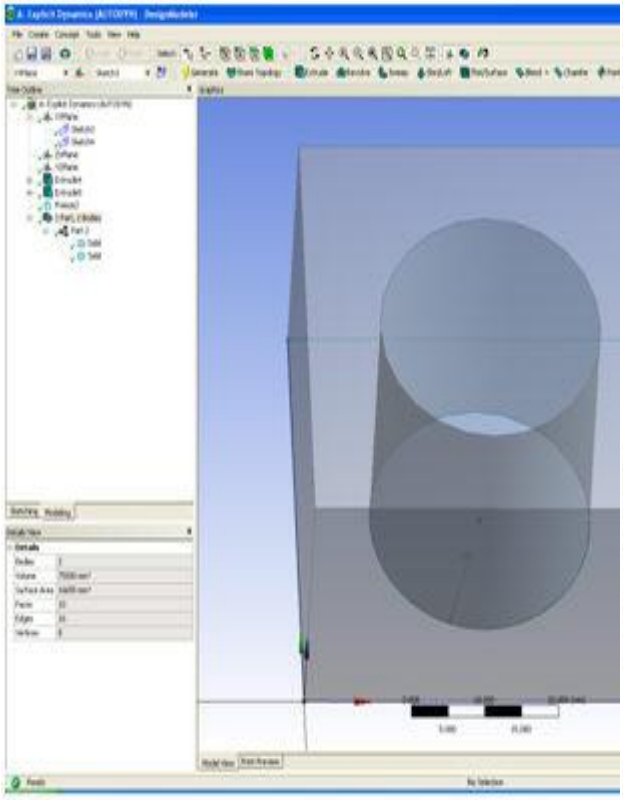
Example use of Delete Elements Group Operation

Split Nodes

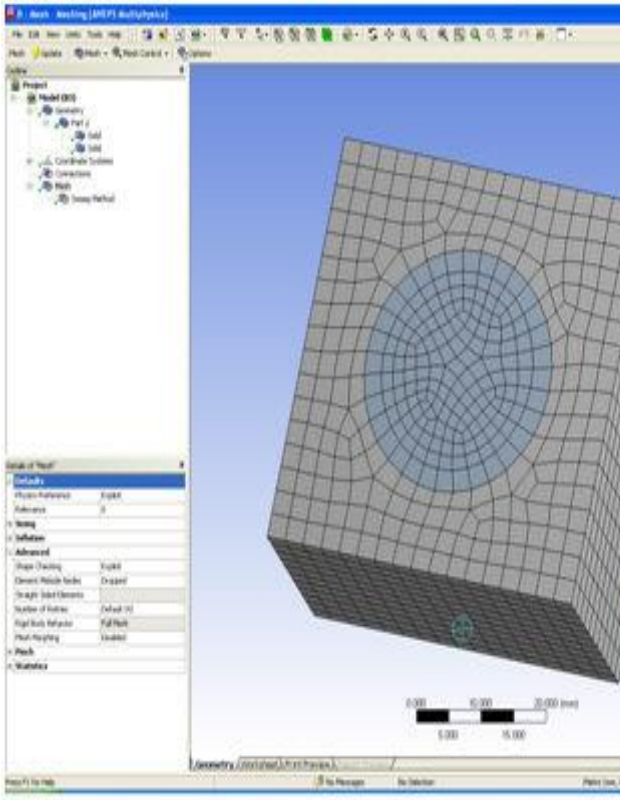
Use this option to split nodes across material boundaries within a selected group. AUTODYN will search through all elements in the selected group and identify nodes that are connected to elements (through the connectivity table) with different materials. Each node connected to more than one material will be replicated by the number of connected materials, while retaining the same physical location. The connectivity of each associated element will be updated.

The option is useful for generating a sliding interface between two materials and enforcing a conforming mesh (coincident nodes) between the material interface at the start of the simulation.

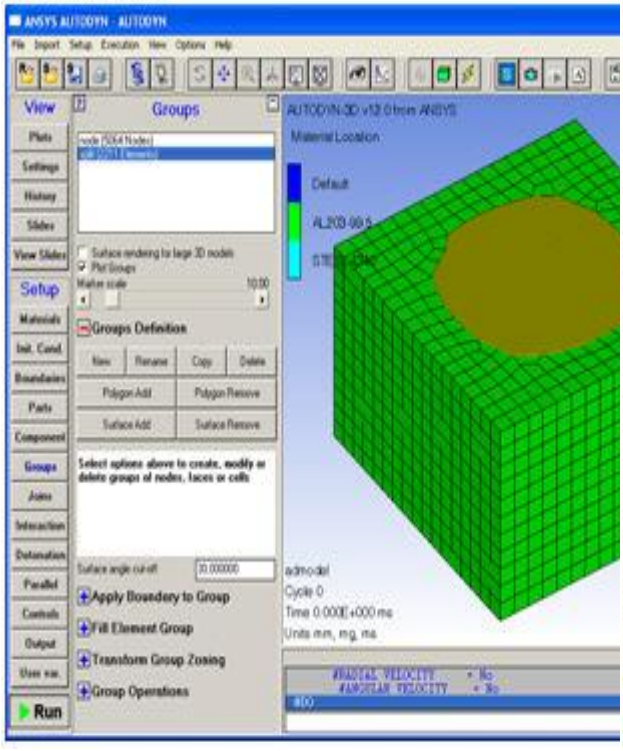
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



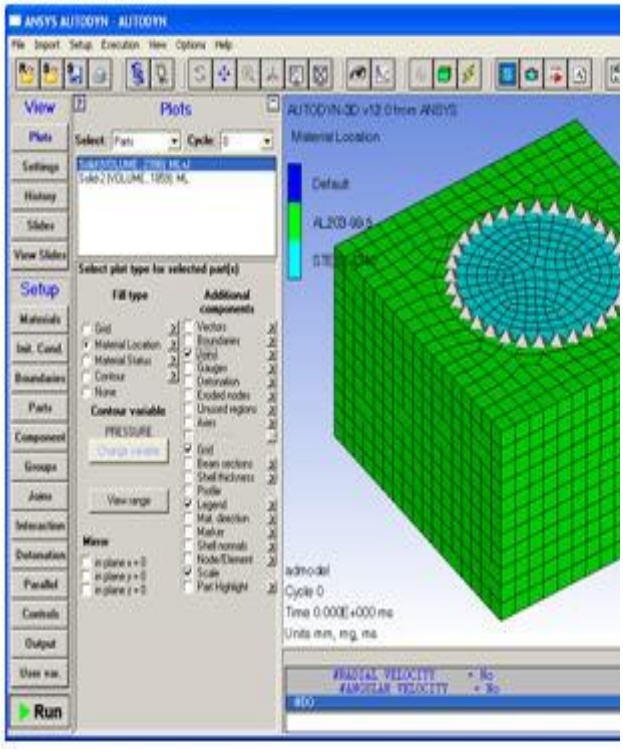
Form Multi-body Part in Design Modeler



Generate Conformal Mesh in Meshing
(with single node at material interface)



Create Element Group in AUTODYN.
Split nodes across material interface.



Nodes are duplicated at material interface to allow sliding contact.

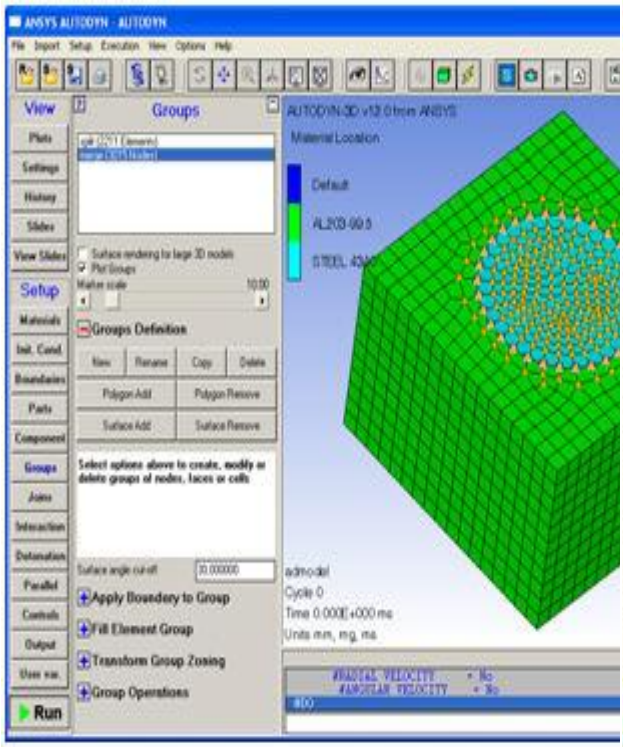
Example use of Split nodes to create conformal sliding interface

Merge Nodes

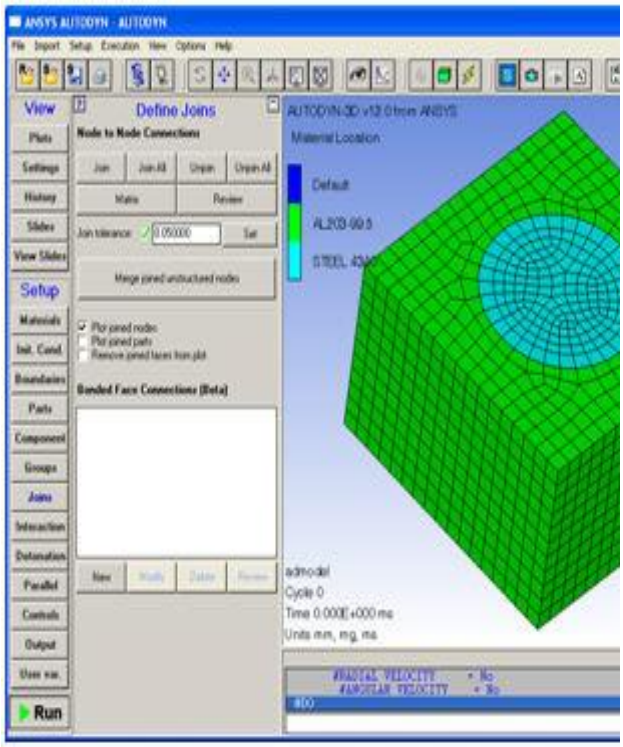
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Use this option to merge coincident (joined) unstructured nodes into a single unstructured node. This option is similar to using the "Merged joined nodes" option on the Join panel; with this option only the nodes that are part of the node group will be merged.

This option will increase the robustness in many applications involving joins.



Create Node Group in AUTODYN.
Merge duplicate nodes.



Duplicate nodes are merged into single nodes.

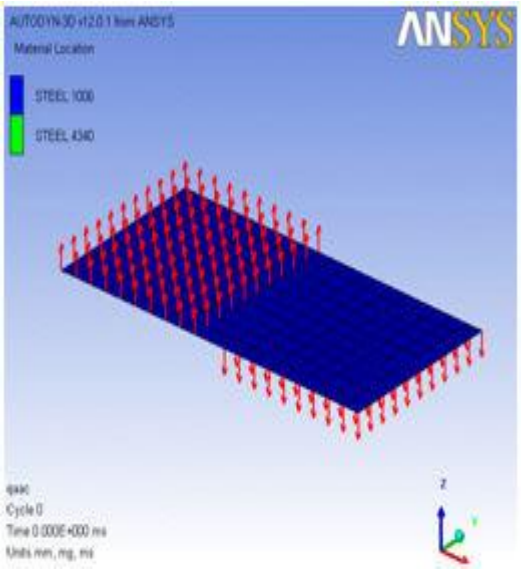
Example use of Merge nodes to remove joins.

Reverse Shell Normals

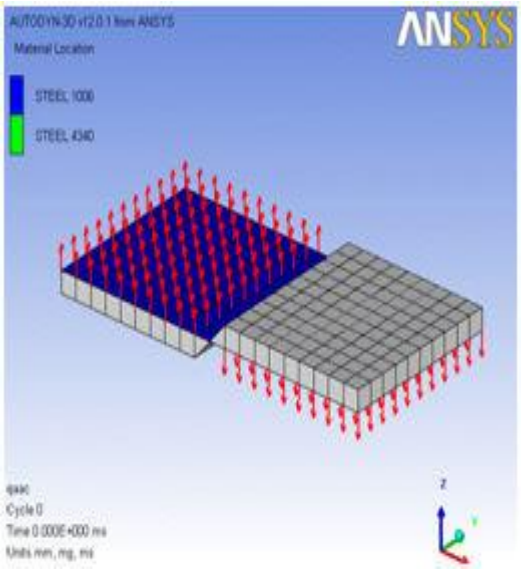
Use this option to reverse the shell element normal of a selected group of shell elements.

This option has to be used when not all element normals of a shell part point in the same direction and Euler-Lagrange coupling is used with thickened shells.

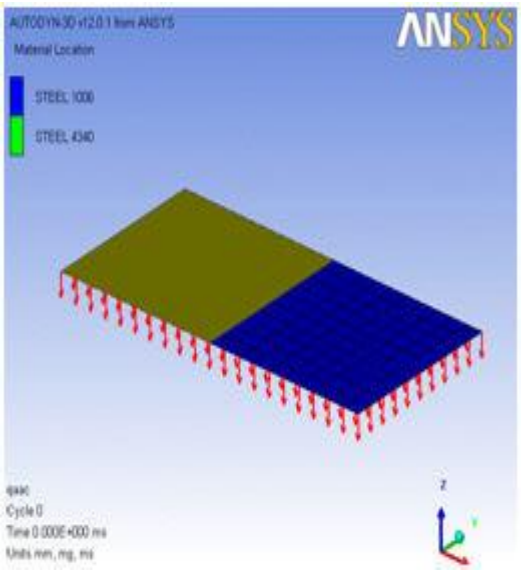
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



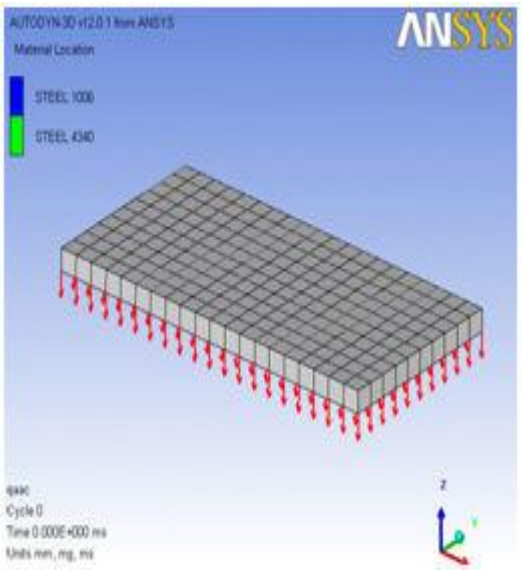
Single part with normals differently directed



Thickened shell provides incorrect coupling surface



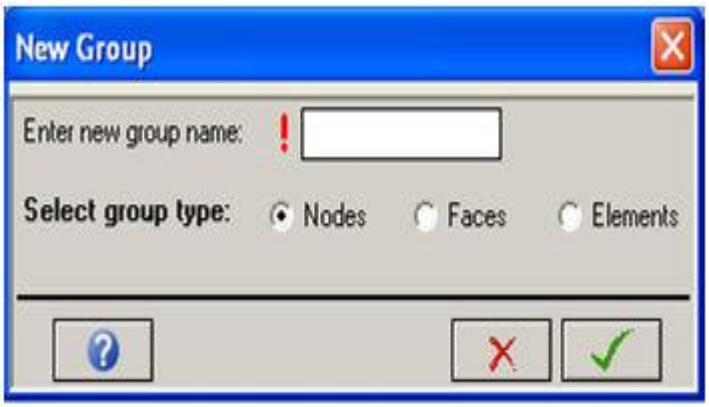
Create element group in AUTODYN Reverse normals



Thickened shell provides correct coupling surface

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups - New



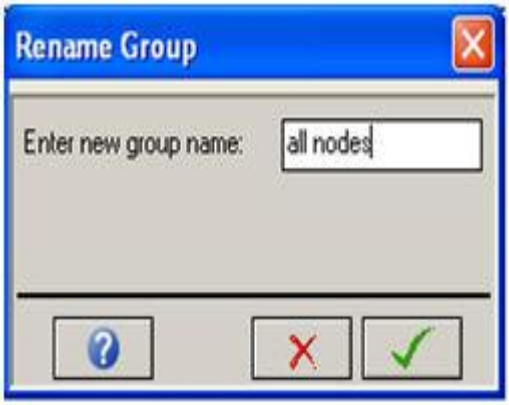
This window lets you create a Group

Group Name
Enter the Group name in this field

Group Type
Select either a node, face or element Group type.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups - Rename

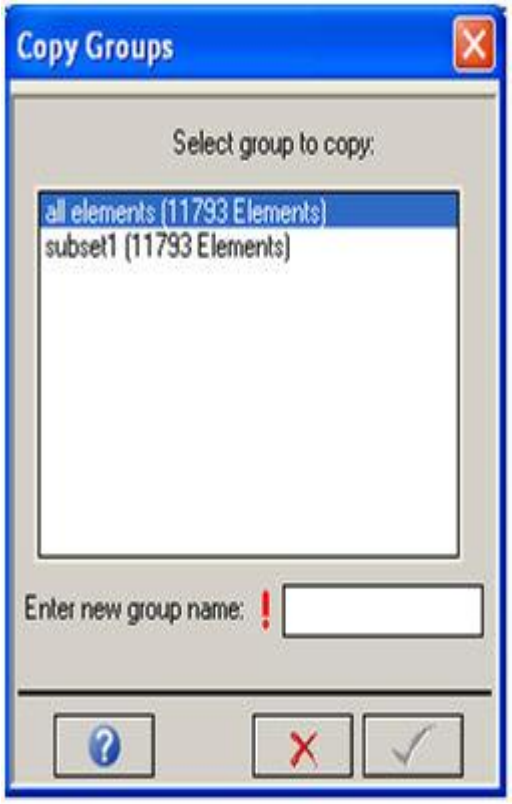


This window lets you rename an existing Group.

Group Name
Enter the new Group name in this field.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups - Copy



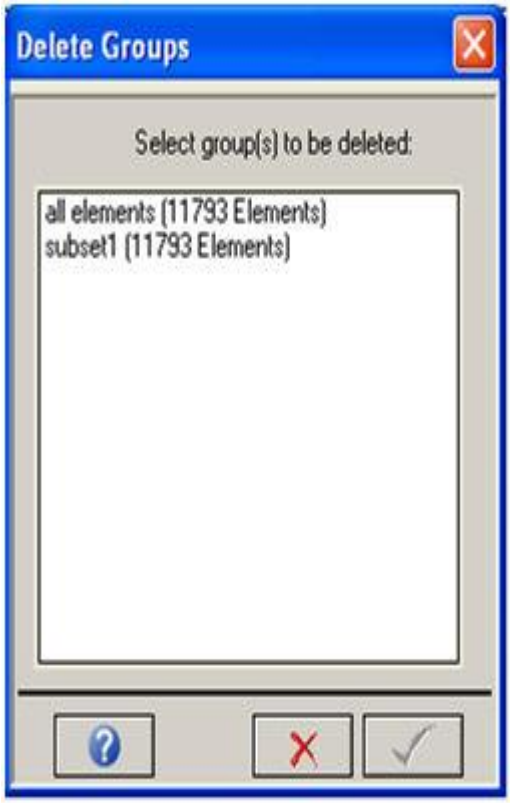
This window lets you create a copy of an existing Group.

Group List
Select the Group that you wish to duplicate

Group Name
Enter the new name for the copied Group

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups - Delete

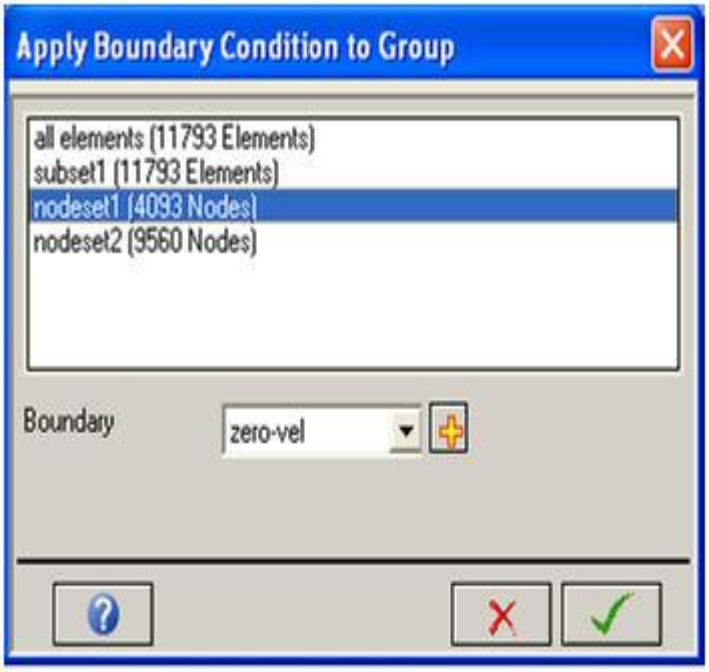


This window lets you delete one or more Groups.

Select the Groups you want to delete in the list and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups - Apply



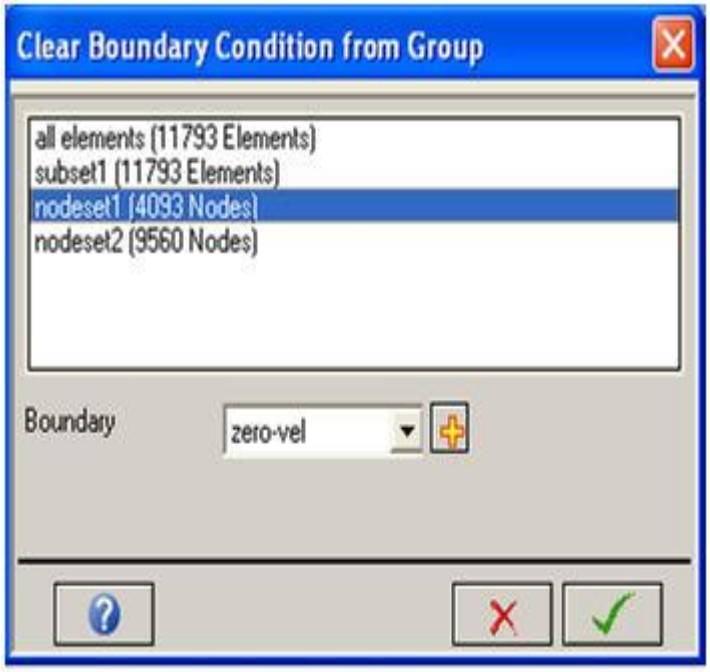
This window lets you apply a boundary condition to a Group. The type of Group and type of boundary condition must be consistent, i.e. you can only apply a nodal boundary condition to a node Group, and a face boundary condition to a face Group.

Group List
Select the Group you wish to apply the boundary condition to.

Boundary
Select the boundary condition to be applied to the Group.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups - Clear



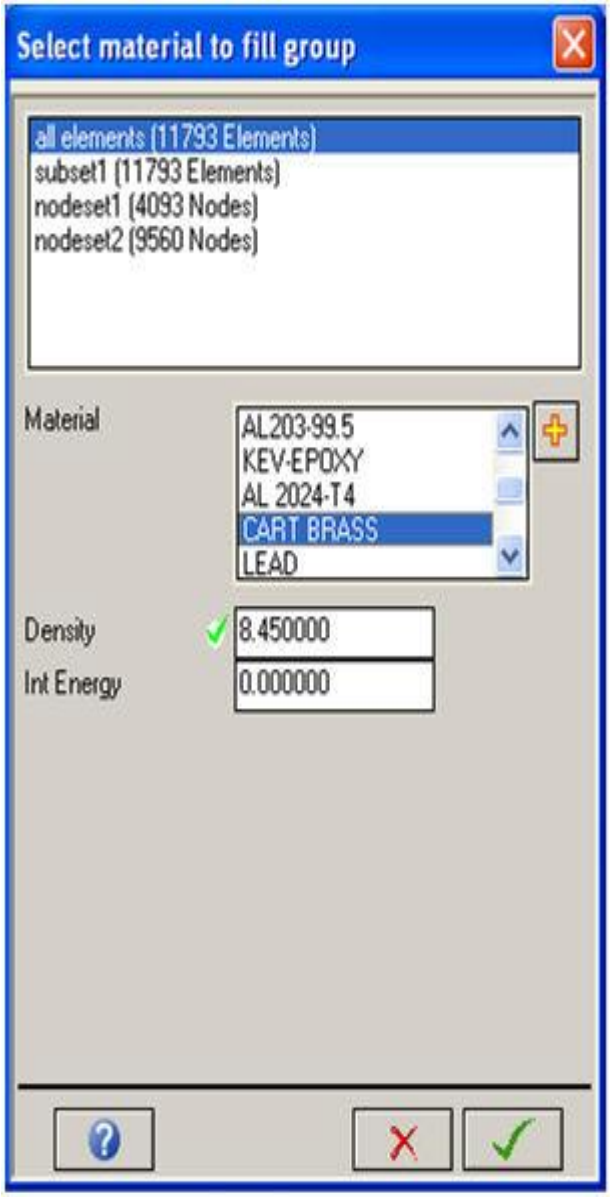
This window lets you clear a boundary condition to a Group. The type of Group and type of boundary condition must be consistent, i.e. you can only clear a nodal boundary condition from a node Group, and a face boundary condition from a face Group.

Group List
Select the Group you wish to clear from the boundary condition to.

Boundary
Select the boundary condition to be cleared from the Group.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups - Material



This window lets you fill the selected element Group with a material.

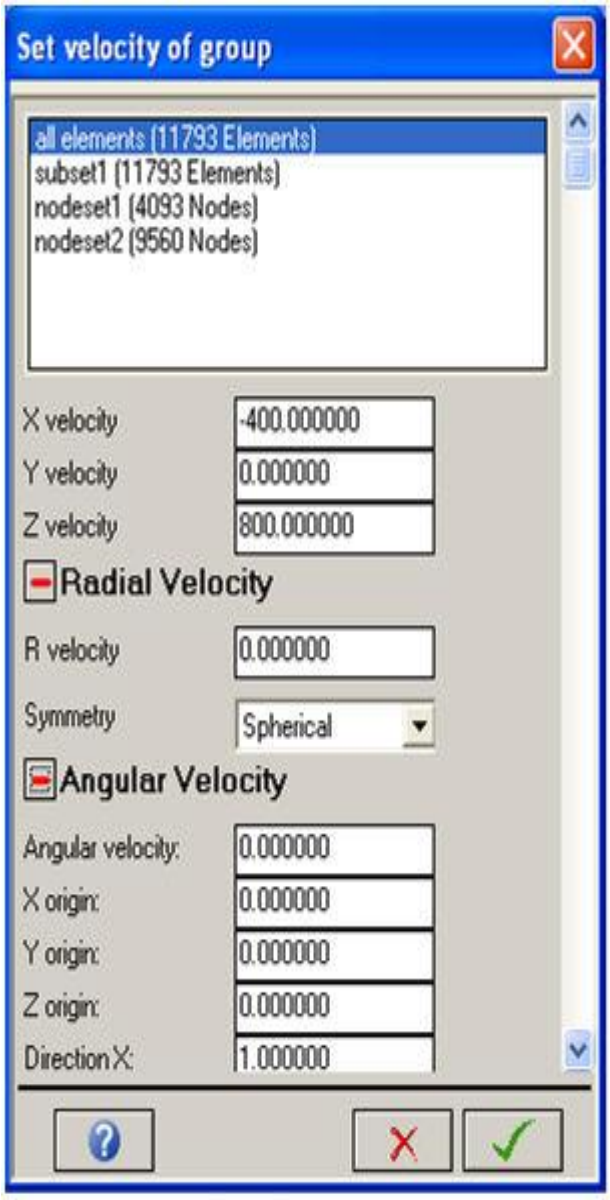
Material
Select the material you want to use to fill your Group from this Pull-down menu.

You can define new materials by press the **Materials** button on the Navigation Bar or by pressing next to the option in this window.

Material Parameters
Enter material parameter values for your fill in the fields provided (Density, Energy).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups-Velocity

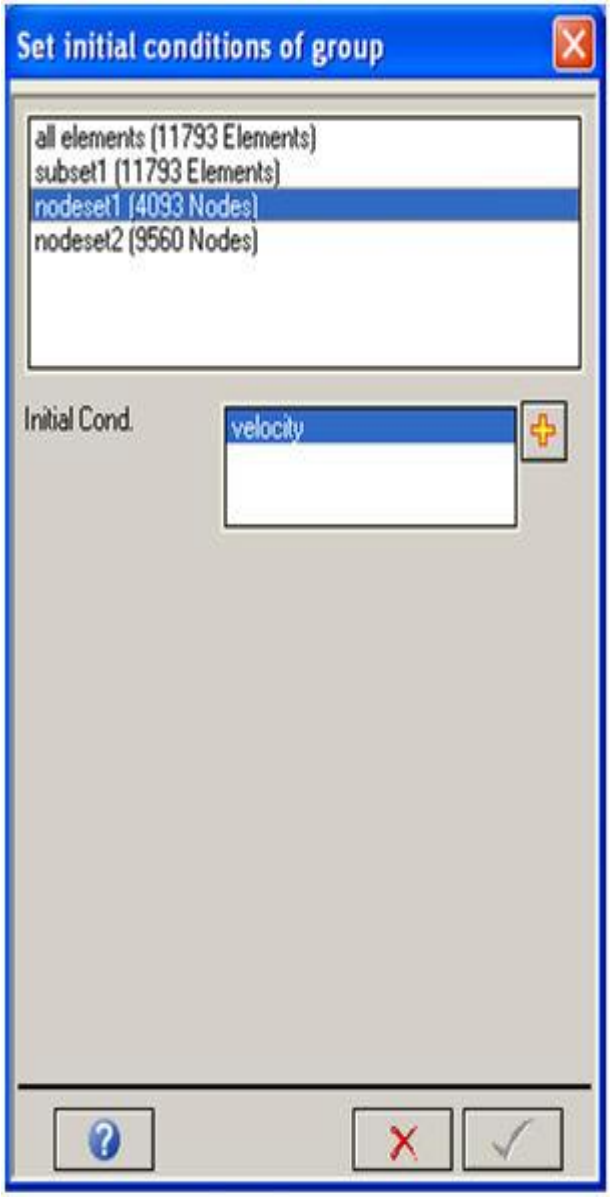


This window lets you assign a velocity to the selected element Group.

Enter the velocity components in the fields provided and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups - Initial Condition



This window lets you assign initial conditions from an Initial Condition Set to the selected element Group.

Fill with Initial Condition Set

Check this box to fill the Group using an Initial Condition Set.

The advantage of filling using an Initial Condition Set is that if you later change one or more parameters in the set, you do not have to refill.

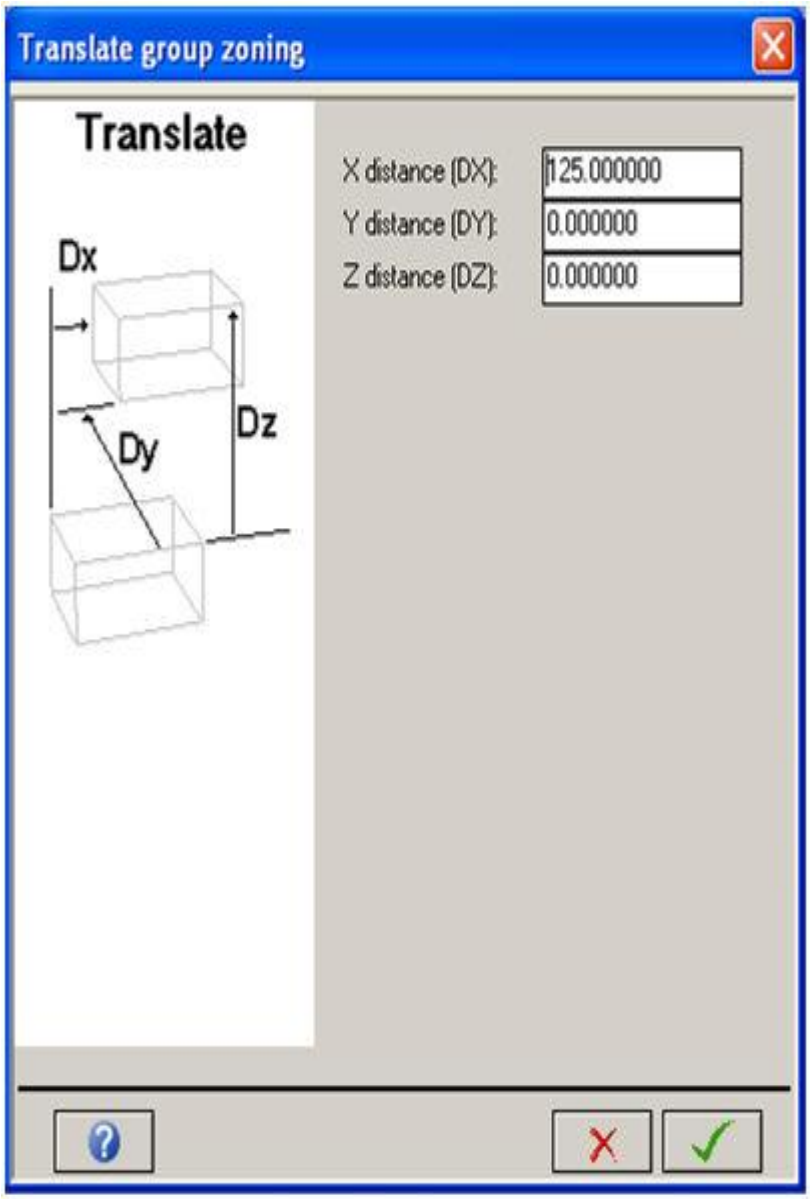
Use the pull-down menu to select the Initial Condition Set you want to use.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

You can define new Initial Condition Sets by pressing the **Init.Cond.** button on the Navigation Bar or by pressing next to the option in this window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups-Translate



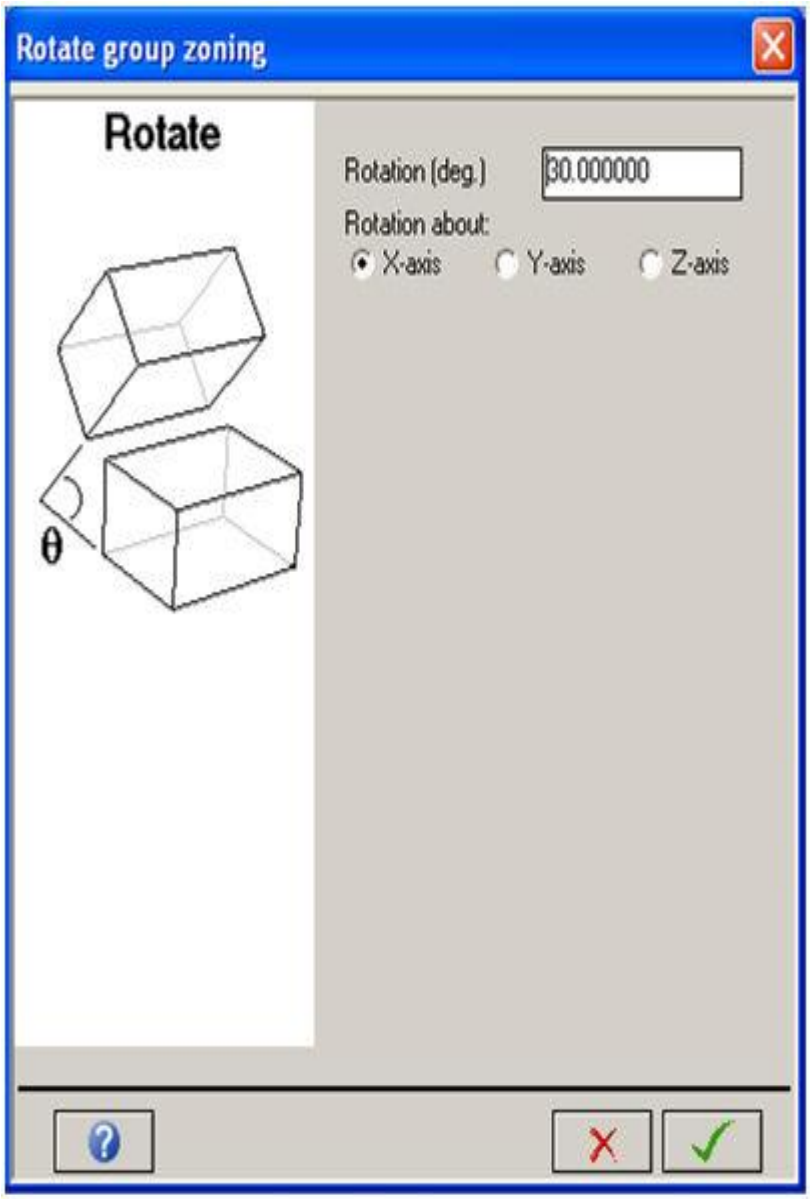
This window lets you translate all nodes associated with the current group in XYZ space.

X/Y/Z distance

Input the X, Y and Z components of the translation in these fields.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups-Rotate



This window lets you rotate all nodes associated with the current group about an axis in XYZ space.

Rotation

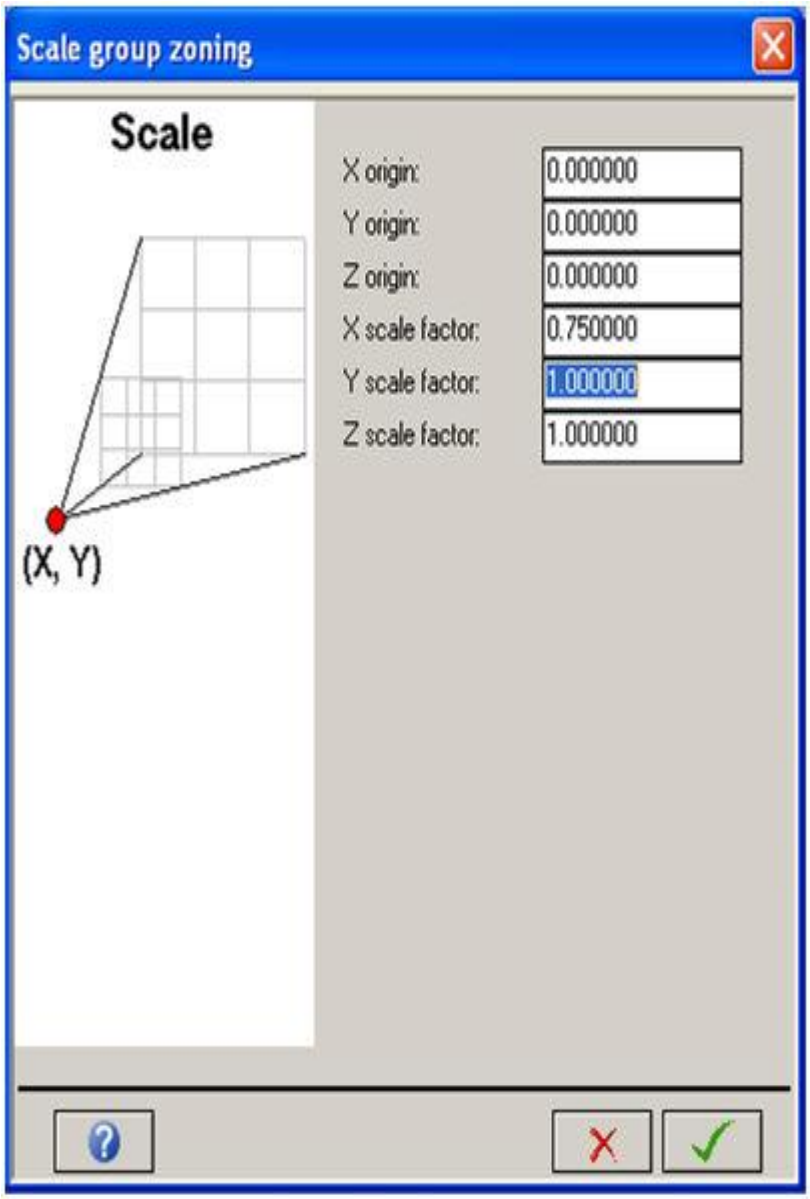
Input the rotation angle in degrees.

X/Y/Z-axis

Specify the axis about which the nodes will be rotated.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Groups-Scale



This window lets you scale all nodes associated with the current group relative to an origin in XYZ space.

Origin

Input the X, Y and Z coordinates of the origin for the scaling.

Scale Factors

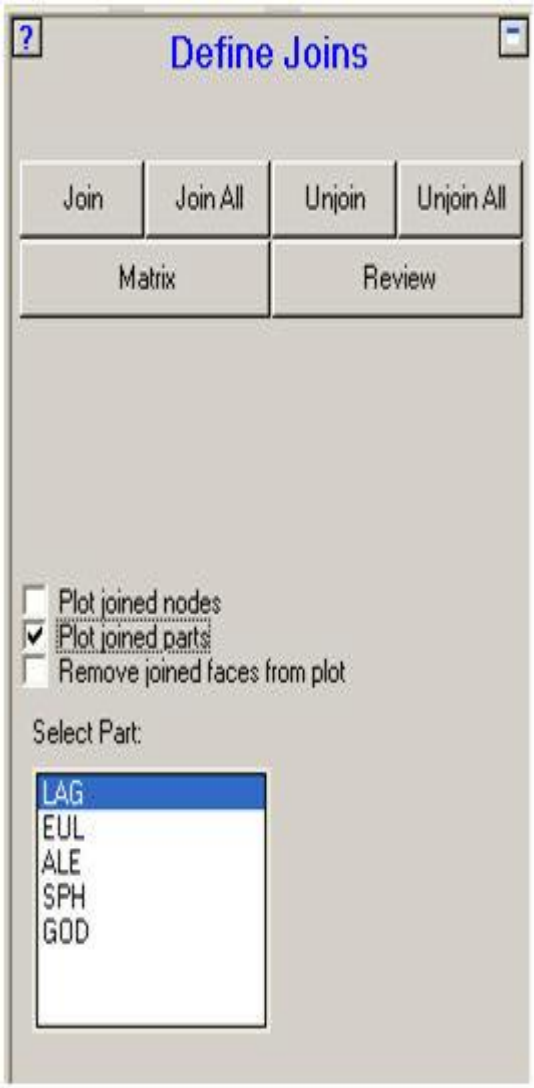
Input the X, Y and Z scale factors.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

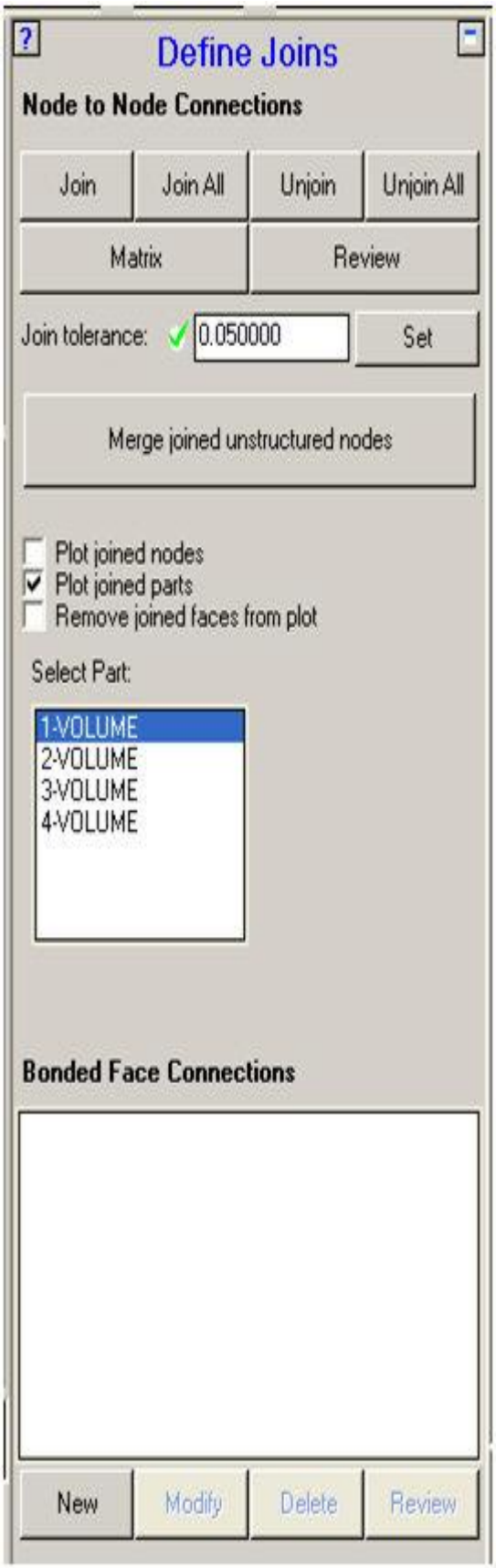
Joins

Joins

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



2D Window



3D Window

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This panel lets you join **Parts** of your model.

If two Parts are joined, AUTODYN automatically finds and joins all coincident nodes in the two parts.

When unstructured Parts are joined the coincident nodes can be merged into a single unstructured node. This option will increase the robustness in many applications involving joins.

Using the Bonded Face Connections option allows the nodes of an unstructured Part to be bonded to a face of another unstructured Part. Bonded contact alleviates the requirement that for joining purposes two nodes need to reside at the same physical location. The nodes that are to be bonded to a face may have an offset with respect to the face and the offset is a user-defined maximum physical normal distance of a node from a face which can be bonded together.

Join

Press this button to join Parts.

Join All

Press this button to join all Parts.

Unjoin

Press this button to unjoin Parts.

Unjoin All

Press this button to unjoin all Parts.

Matrix

Press this button to defined join Parts using a matrix.

Review

Press this button to review which Parts are currently joined.

Join tolerance

Enter the tolerance you want AUTODYN to use in determining which nodes should be joined. Press the **Set** button to set this tolerance.

The default value for the join tolerance will take care of small round-off differences in node location and will give good results. The tolerance should not artificially be increased to try to join nodes with a clear offset with respect to each other.

Merge joined nodes

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select this option to merge joined unstructured nodes into a single unstructured node.

Plot Joined Nodes

Check this box to plot a symbol at every joined node.

Plot Joined Parts

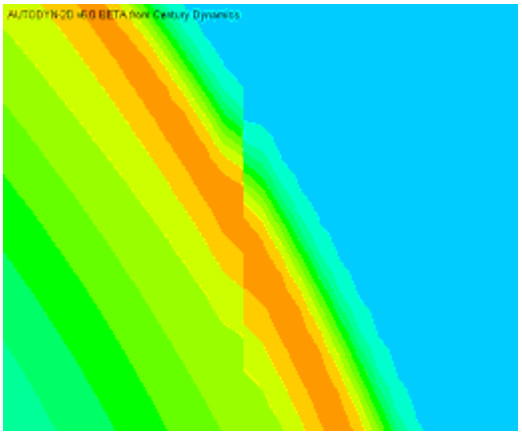
Check this box to show which Parts are joined to a selected Part (a box appears to let you select a Part).

Remove joined faces from plot

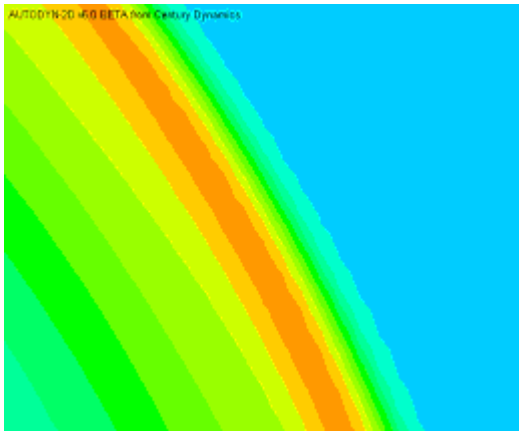
Selecting this option results in continuous contours across joined faces and also removed joined faces from other types of plots.

Improved Rendering Across Joined Nodes

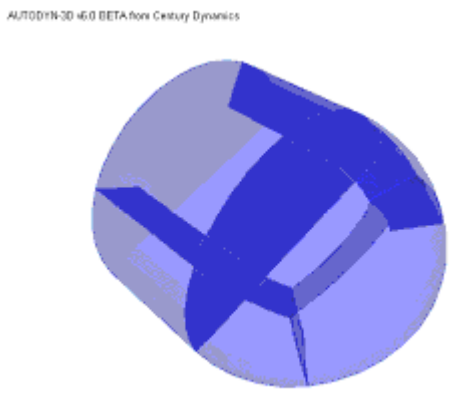
There is now an option to remove joined faces from rendered images under the Joins, Settings option. Selecting this option results in continuous contours across joined faces and also removed joined faces from other types of plots.



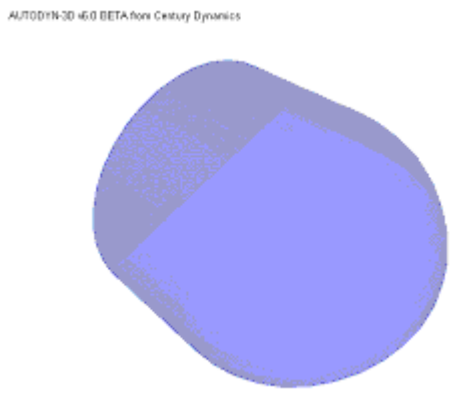
Contour plot including joined faces



Contour plot with "remove joined faces" set



Material plot using opacity



Material plot using opacity with "remove

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

joined faces" set.

Bonded Face Connections

Bonded contact allows the nodes of an unstructured Part to be bonded to a face of another unstructured Part. Bonded contact alleviates the requirement that for joining purposes two nodes need to reside at the same physical location. The nodes that are to be bonded to a face may have an offset with respect to the face and the offset is a user-defined maximum physical normal distance of a node from a face which can be bonded together.

Bonded face connections can fail and break upon a user-defined stress criterion.

Spot Welds

Spot welds generated in an ANSYS Explicit Dynamics systems are transferred to the AUTODYN system as beam elements filled with a rigid material.

Spot welds utilise the multi-body rigid material option where each spot weld acts as an individual rigid boy.

Spot welds can be defined as breakable by force:

- Normal force limit: maximum normal force that can be sustained by the weld
- Shear force limit: maximum shear force that can be sustained by the weld.

Failure of the spot weld occurs when:

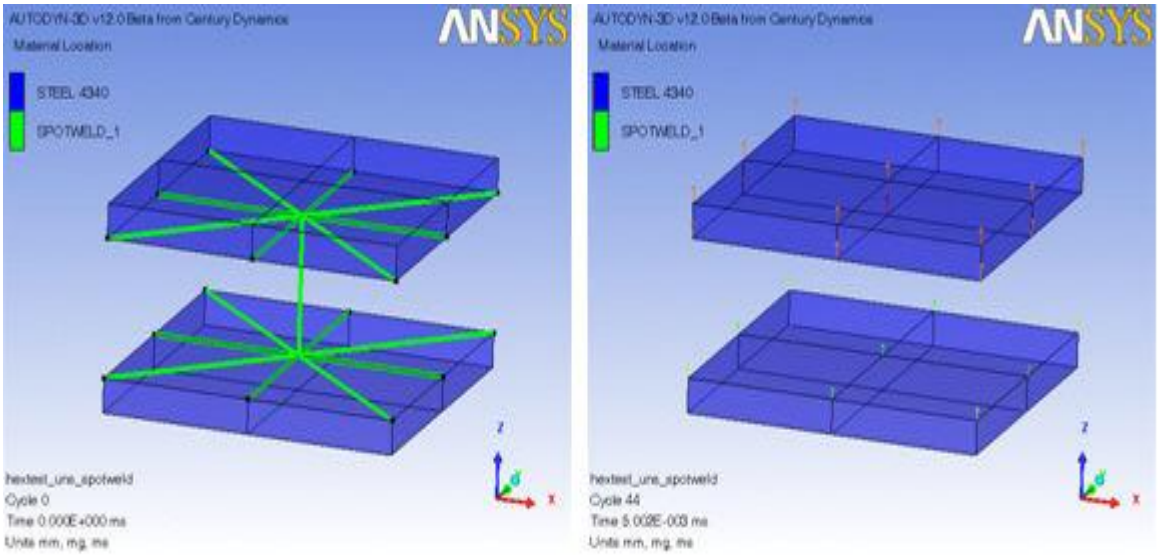
$$\left(\frac{|f_n|}{S_n}\right)^{\exp n} + \left(\frac{|f_s|}{S_s}\right)^{\exp s} \geq 1$$

where f and f are normal and shear interface forces, S and S are the maximum allowed normal and shear force limits and *expn* and *exps* are user defined exponential coefficients. The normal interface force f is nonzero for tensile values only.

Note that spot welds cannot be generated directly inside an AUTODYN system. Spot welds should be defined in an ANSYS Explicit Dynamics system and automatically transferred to a linked AYTODYN system.

For each spot weld connections a multiple beam rigid body is automatically generated for each spot weld in the ANSYS Explicit Dynamics system in order to guarantee the transfer of bending over the spot weld.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



After failure of the spot weld the spot weld rigid body will be removed from the analysis model as can be seen above.

Within an ANSYS Explicit Dynamics system an effective area can be input for spot welds, thus the breakable criteria can be defined in terms of stress. This stress is converted to force when transferred into AUTODYN.

Spot welds of zero length are permitted. However, if such spot welds are defined as breakable the above failure is modified since local normal and shear directions cannot be defined. A modified criterion using global forces is used instead:

$$\sqrt{(df_x^2 + df_y^2 + df_z^2)/(s_n^2 + s_s^2)} < 1.0$$

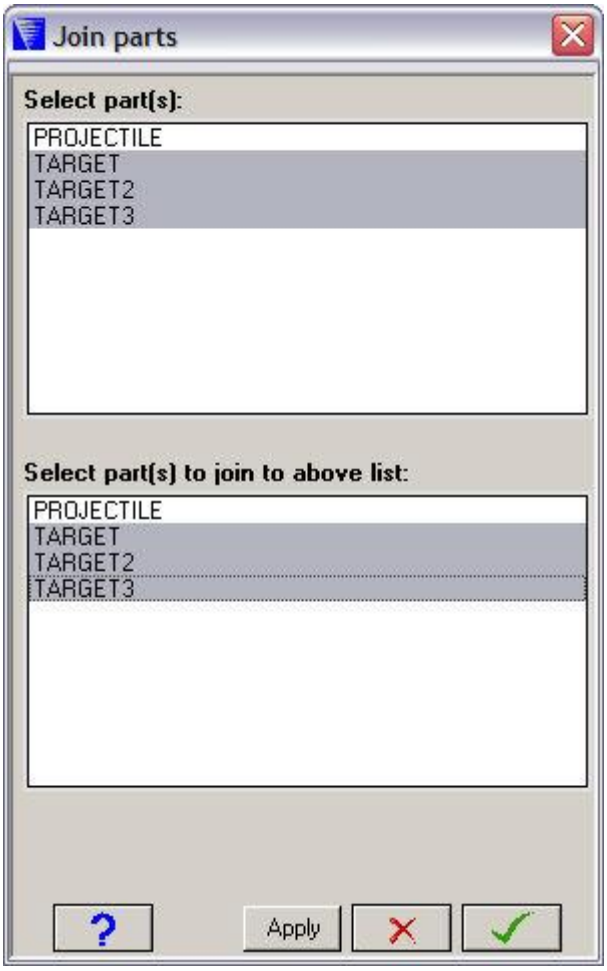
where dfx, dfy and dfz are the global force differences across the spot weld.

Notes:

1. A spot weld is equivalent to a rigid body and as such multiple nodal boundary conditions cannot be applied to spot welds.
2. Currently analysis models containing spot welds cannot be run in parallel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Joins - Join



This window lets you join Parts.

This method of joining Parts works well no matter how many Parts you have in your model.

If you only have a few Parts, you may prefer to use a **Matrix** to choose the Parts you want to join.

Select Part(s)

This box lists all the Parts in your model. Select one or more Parts in this list.

Select Part(s) to join to the above list

This box also lists all the Parts in your model. Select one or more Parts in this list.

Apply

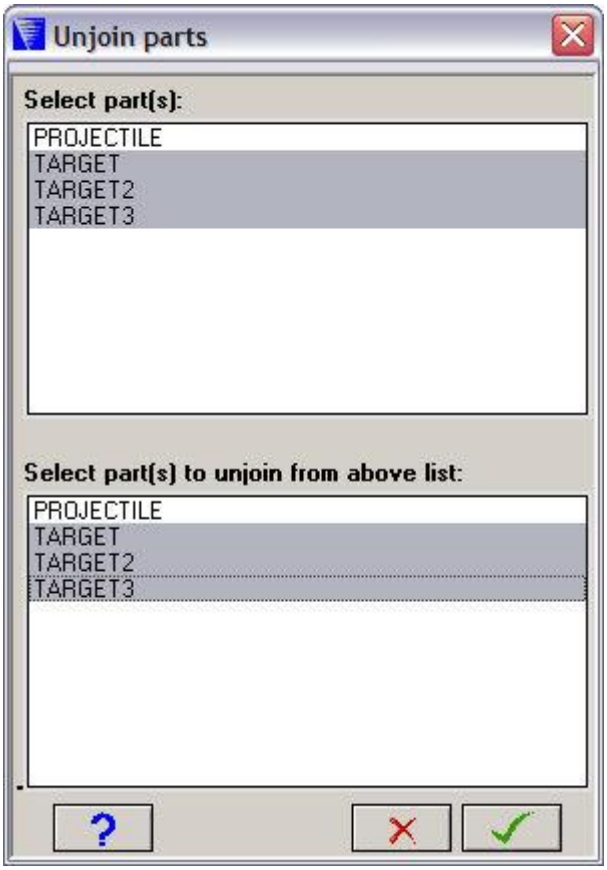
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press this button to join all Parts selected in the top window with all Parts selected in the lower window.

You can apply the above procedure as many times as you wish without exiting this window. Every time you press **Apply**, the specified Parts are joined.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Joins - Unjoin



This window lets you unjoin Parts.

This method of unjoining Parts works well no matter how many Parts you have in your model.

If you only have a few Parts, you may prefer to use a **Matrix** to choose the Parts you join (or unjoin)

Select Part(s)

This box lists all the Parts in your model. Select one or more Parts in this list.

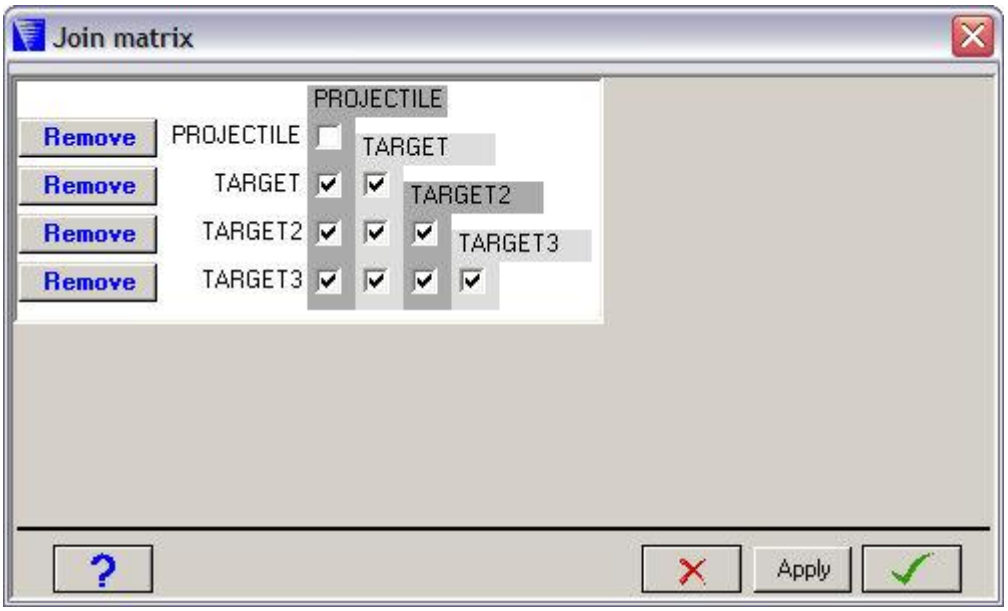
Select Part(s) to unjoin to the above list

This box also lists all the Parts in your model. Select one or more Parts in this list.

Press ✓ to unjoin all Parts selected in the top window with all Parts selected in the lower window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Joins - Matrix



This window lets you choose the Parts you want to join by checking boxes in a matrix.

If you have a large number of Parts, you may find it easy to join / unjoin Parts using the **Join** / **Unjoin** options in the Joins dialog panel.

The matrix shown in this panel list the Parts of your model vertically and horizontally. This matrix is symmetrical, so only the diagonal and lower part of the matrix is shown.

Check a box to join the two associated Parts.

The Remove buttons let you quickly uncheck the boxes for each row of the matrix.

Press

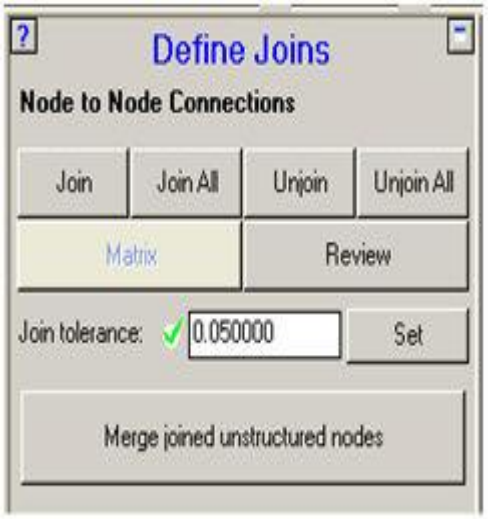
to apply the changes you make to the matrix.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Merge Joined Nodes

The option of merging joined unstructured nodes will merge the (multiple) joined nodes that reside at the same physical location in the model into one single unstructured node.

Since trajectory contact does not allow the use of joined unstructured nodes in the contact, the merge option can be used to remove joins from the model. Bonded connections are recommended over node to node joins for unstructured models.



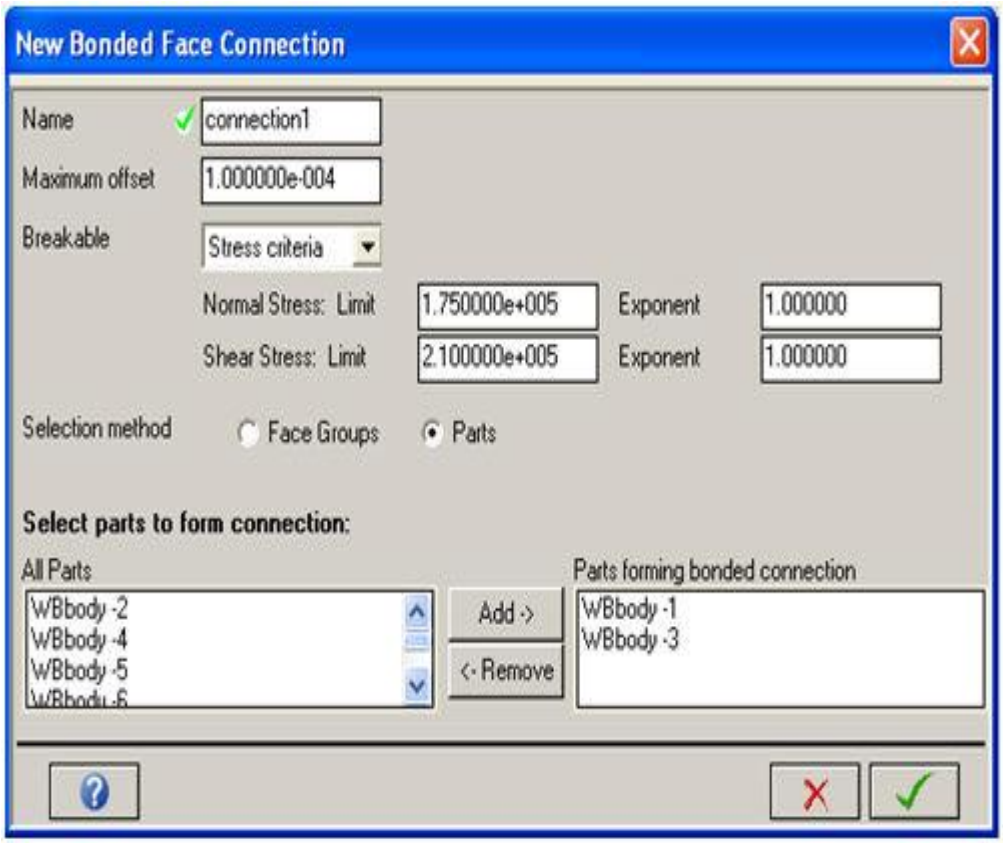
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Bonded Face Connections

Bonded contact allows the nodes of an unstructured Part to be bonded to a face of another unstructured Part. Bonded contact alleviates the requirement that for joining purposes two nodes need to reside at the same physical location. The nodes that are to be bonded to a face may have an offset with respect to the face and the offset is a user-defined maximum physical normal distance of a node from a face which can be bonded together.

During the analysis the nodes are kept at the same relative position on the face to which they are bonded.

Bonds are allowed to fail and break upon a user-defined criterion.



Name

Enter a name for the bonded face connection set.

Maximum Offset

The automatic search algorithm for bonded regions will bond a node to the first face found within the specified offset distance. To ensure nodes are bonded to the nearest face, the value of the offset distance should be defined to be as small as possible.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Breakable

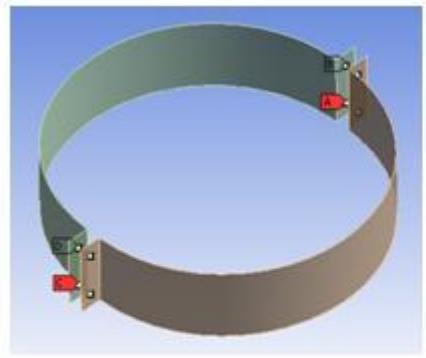
By default the bonded face connection will not be breakable. When the option Stress criteria is selected the bonded connection will fail and break upon a user-defined stress failure criterion:

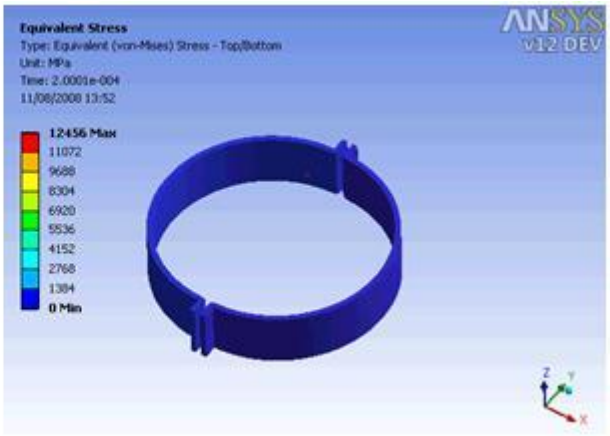
- By stress:
  - Normal stress limit: maximum normal stress that can be sustained at the bond
  - Shear stress limit: maximum shear stress that can be sustained at the bond

Failure of the bond occurs when:

$$\left(\frac{|\sigma_n|}{S_n}\right)^{\exp n} + \left(\frac{|\sigma_s|}{S_s}\right)^{\exp s} \geq 1$$

where σ_n and σ_s are normal and shear stress at the bond S_n and S_s are the maximum allowed normal and shear stress limits and *expn* and *exps* are user defined exponential coefficients. The normal stress at the bond σ_n is nonzero for tensile values only.





Selection method

Bonded joins can be generated in AUTODYN on a part basis as well as on a (multiple) face group basis.

Bonded Connections can also be defined in an Explicit Dynamic system and automatically transferred into a linked AUTODYN system.

Notes

- The element variable BOND_STATUS is used to record how many nodes are actively bonded to the faces of an element. This variable can be contoured to verify that bonds are correctly defined. If a bond fails during the simulation the bond status will show a reduction in the number of active bonds. When an element has no remaining bonds the value of BOND-STATUS is set to -1.0.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

- Bonded contact is only available with trajectory contact.
- Bonded contact is available for unstructured volume, shell and beam elements.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions

Interactions

The Interactions dialog panel lets you define interactions between the various Parts of your model.

Buttons at the top of this panel let you choose which type of interaction you want to set up:

Lagrange/Lagrange

Press this button to set up impact / slide interactions between Lagrangian Parts (Parts using a Lagrange, Shell, Beam or SPH solver).

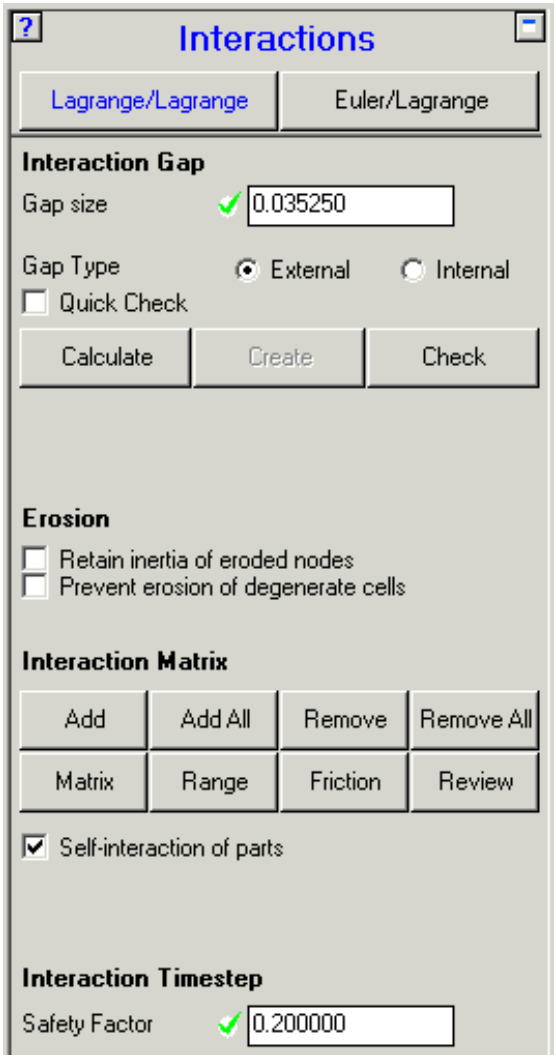
Euler/Lagrange

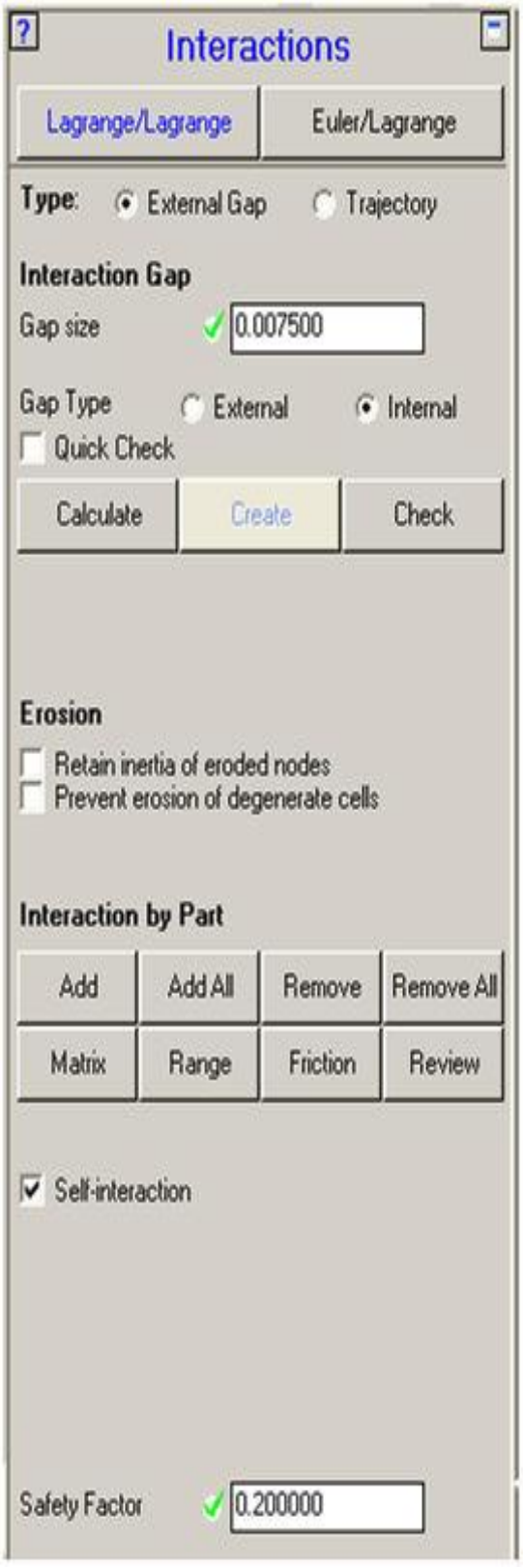
Press this button to set up coupling between Lagrangian Parts and Eulerian Parts.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Lagrange/Lagrange

Interactions - Lagrange / Lagrange





Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

2D 3D

This panel lets you set up interactions between Lagrangian Parts (Parts using a Lagrange, Shell, Beam or SPH solver).

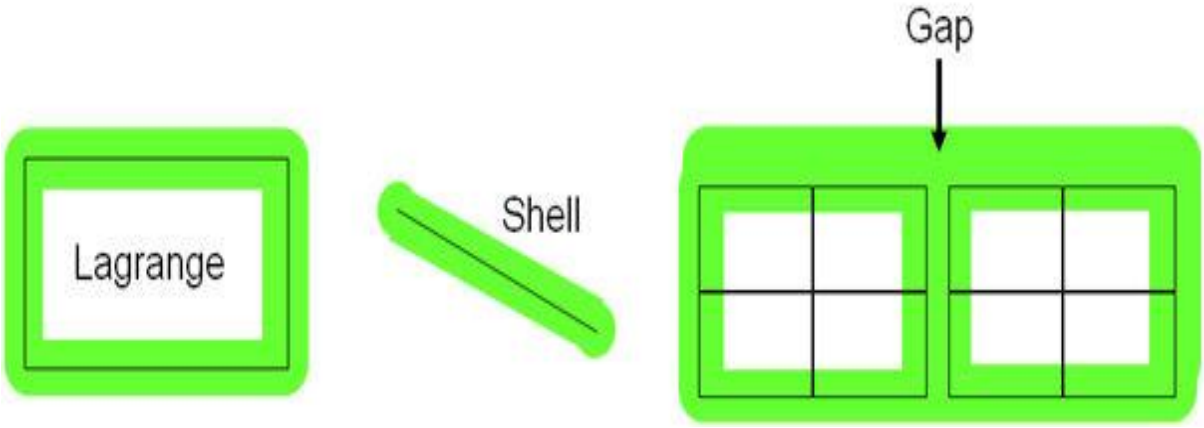
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Lagrange/Lagrange Introduction

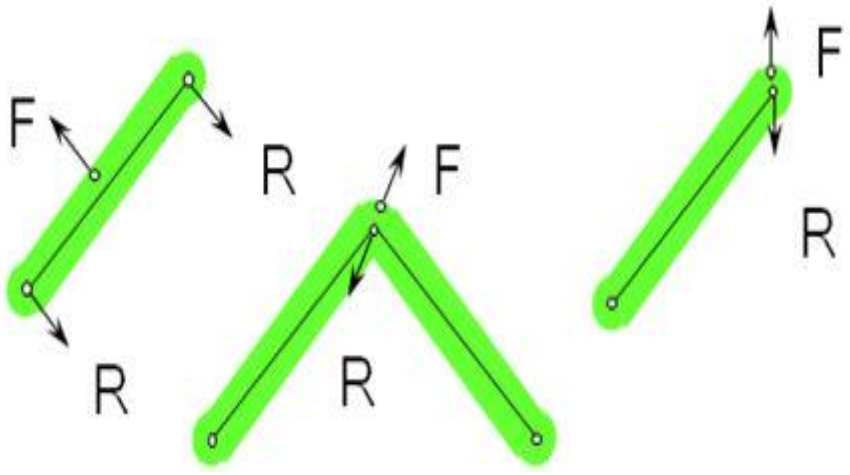
AUTODYN offers 2 types of Lagrange/Lagrange interaction (contact) methods.

Gap contact

In this contact method each surface segment is surrounded by a contact detection zone. The dimensions of this contact zone is called Gap size. This means that initially the outside surfaces of the parts in the contact should be separated by the gap distance.



Any nodes entering the contact detection zone of a surface segment are repelled by a force normal to the segment surface and proportional to the depth of penetration of the node into the contact detection zone.

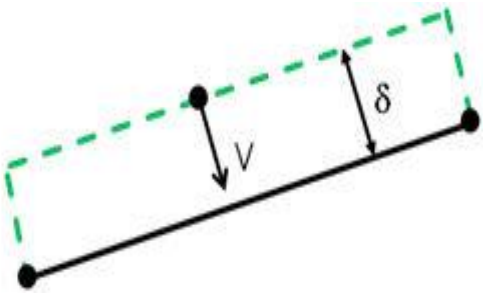


The interaction conserves linear and angular momentum.

To assure a stable interaction process the gap contact algorithm uses a timestep restriction such that during 1 computational time step a surface node cannot travel more than 20% into the contact detection zone.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

$$\Delta t = 0.2\frac{\delta}{V}$$



where δ is the gap size and V the velocity of the penetrating node.

The gap contact is available for all structured as well as unstructured Lagrange Parts, with exception for 2D unstructured Parts.

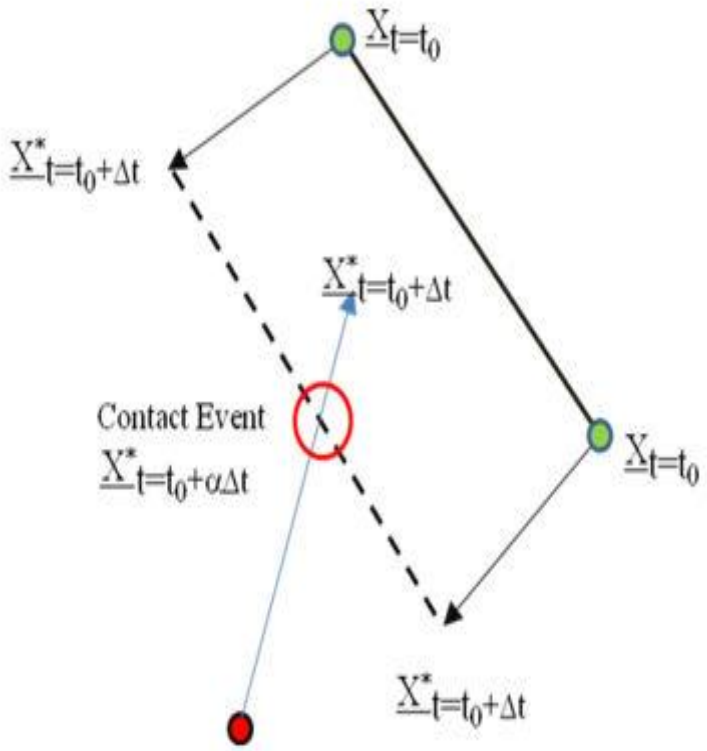
Trajectory contact

The Trajectory contact algorithm is available for all 3D Unstructured solvers, including SPH, and has three main potential benefits over the Gap based contact algorithm.

- There is no requirement to specify a contact detection zone or leave a physical gap between Parts at the start of a simulation. This makes model generation for complex 3D geometries significantly easier.
- There is no constraint on the timestep due to contact. The algorithm detects node to fact contact by tracking the trajectory of the nodes and faces over time. The removal of the timestep constraint on contact can give very significant performance improvements.
- The algorithm is energy conserving, in addition to momentum conserving.

The trajectory of nodes and faces included in frictional or frictionless contact are tracked during the computational cycle. If the trajectory of a node and a face intersects during the cycle a contact even is detected.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



If a contact event is detected the node involved in the contact will be pushed back towards the true contact position during the computational cycle.

There are two methods available:

1. **Penalty**
   If a contact event is detected, a local penalty force is calculated to push the node back to the face. Equal and opposite forces are calculated on the nodes of the face in order to conserve linear and angular momentum. The applied penalty force will push the node back towards the true contact position during the cycle. However, it will take several cycles to satisfy the contact condition. Kinetic energy is not necessarily conserved. The conservation of energy can be tracked using the energy time history.
2. **Decomposition Response (DCR)**
   All contacts that take place at the same point in time are first detected. The response of the system to these contact events is then calculated to conserve momentum and energy. During this process, forces are calculated to ensure that the resulting position of the nodes and faces does not result in further penetration at that point in time. The decomposition response algorithm is more impulsive (in a given computational cycle) than the penalty method. This can give rise to large hourglass energies and energy errors.

The trajectory contact can handle multiple contact of a node with different faces of a contact surface during a computational timestep.

Current Limitations

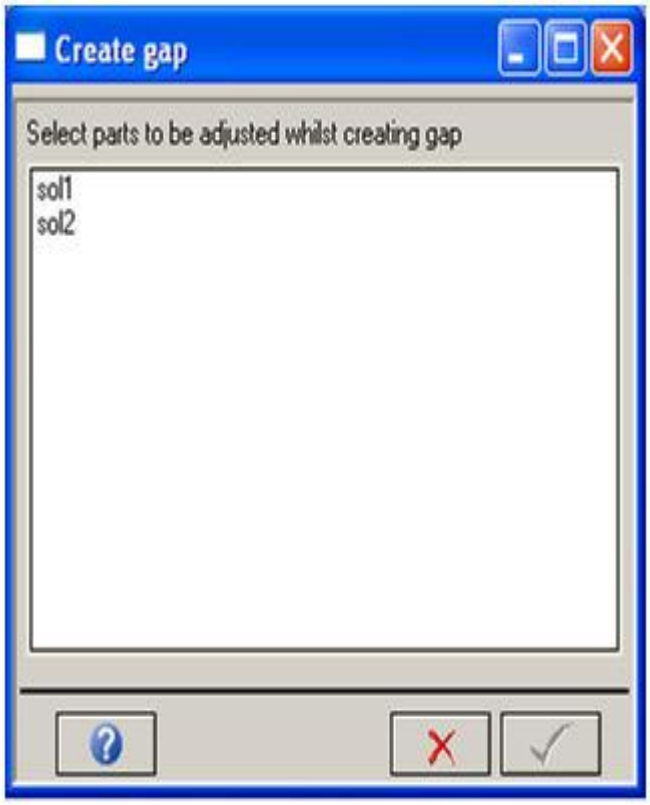
- The trajectory contact option cannot currently be used in parallel simulations.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

- The trajectory contact option cannot be used in combination with joined unstructured parts. To circumvent this limitation the option to merge all joined unstructured nodes can be used.
- Interaction of SPH nodes, accounting for their full natural thickness/size is currently not available.
- The decomposition response algorithm cannot currently be used in combination with bonded contact regions. The formulation will automatically switch to the Penalty method if bonded regions are present in the model.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Lagrange / Lagrange - Create



This window lets you create a gap between structured Parts automatically. This option doesn't work for unstructured parts and therefore unstructured Parts will not appear in the window.

Gaps are created by reducing the size of surface elements of the selected Parts, so there will be small changes in the mass of the Parts that are modified.

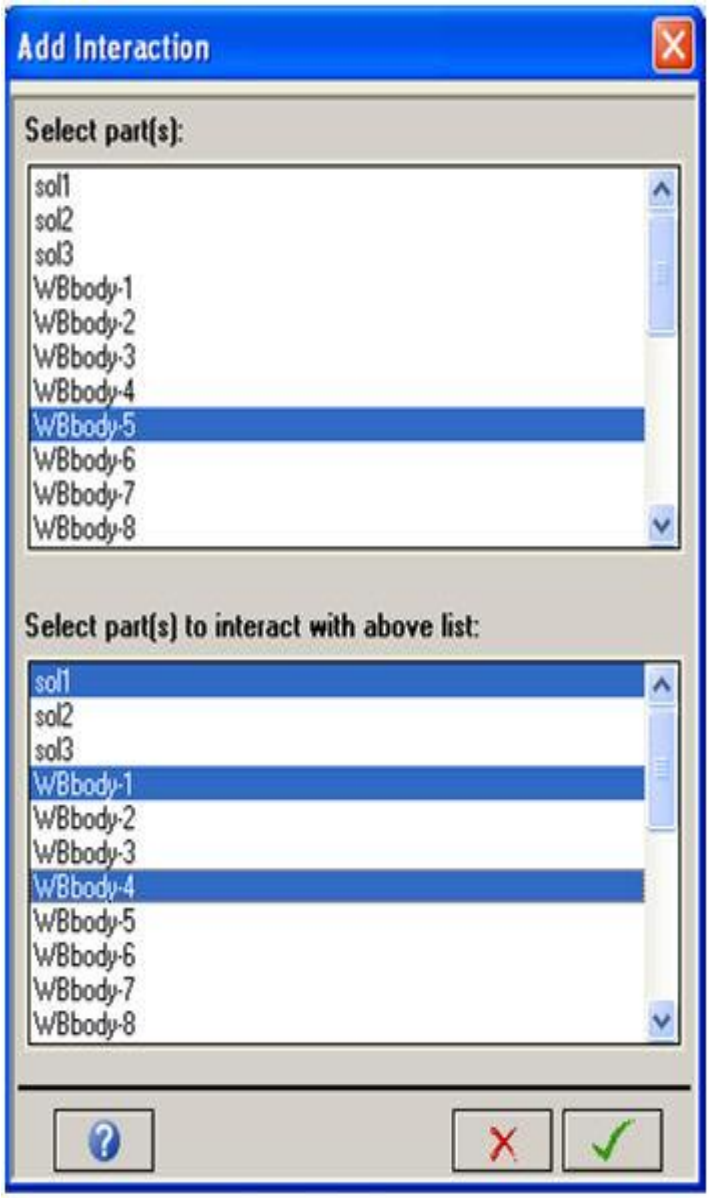
You may prefer to manually inset gaps using the **Translate** and / or **Scale** options in the **Zoning** panel (no mass will be lost).

The box in this window lists all Parts involved in interactions.

Select the Part(s) you want to adjust while creating the gaps and press for AUTODYN to automatically create the gaps.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Lagrange / Lagrange - Add



This window lets you add interactions between one or more Parts.

This method of adding interactions works well no matter how many Parts you have in your model.

If you only have a few Parts, you may prefer to use a **Matrix** to choose the interactions.

Select Part(s)

This box lists all Parts involved in interactions. Select one or more Parts in this list.

Select Part(s) to interact with the above list

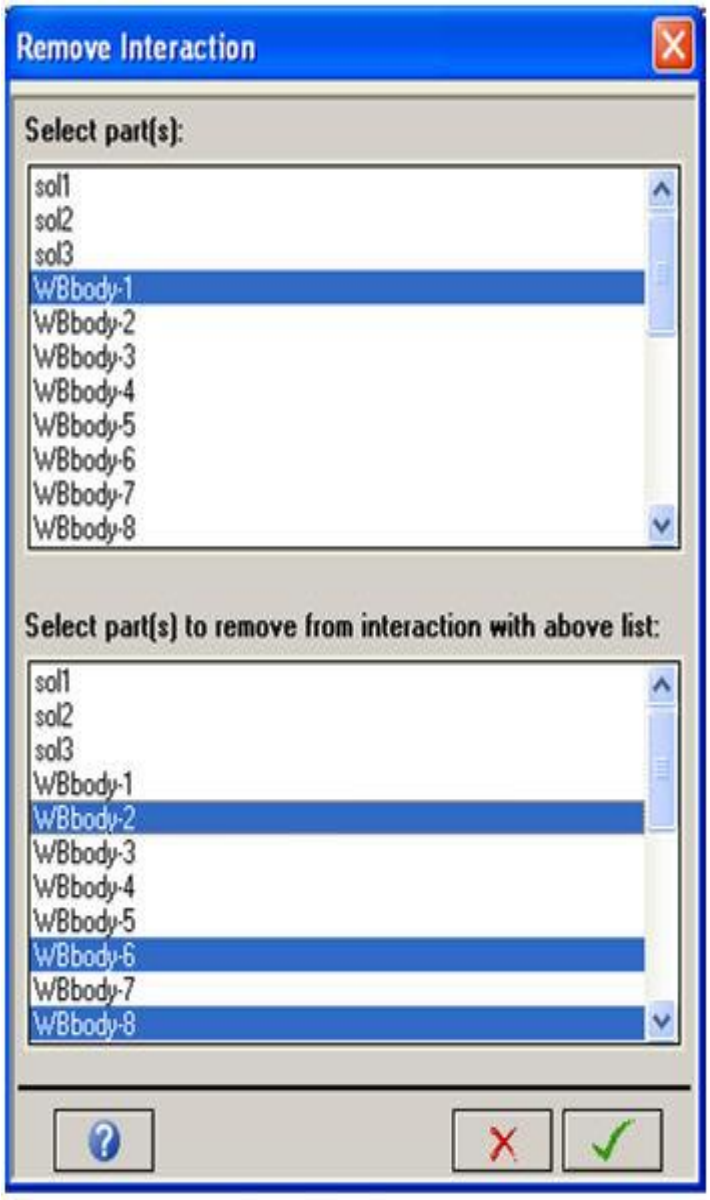
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This box also lists all Parts involved in interactions. Select one or more Parts in this list.

Press to set all Parts selected in the top window to interact with all Parts selected in the lower window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Lagrange / Lagrange - Remove



This window lets you remove interactions between one or more Parts.

This method of removing interactions works well no matter how many Parts you have in your model.

If you only have a few Parts, you may prefer to use a **Matrix** to add or remove interactions.

Select Part(s)

This box lists all Parts involved in interactions. Select one or more Parts in this list.

Select Part(s) to interact with the above list

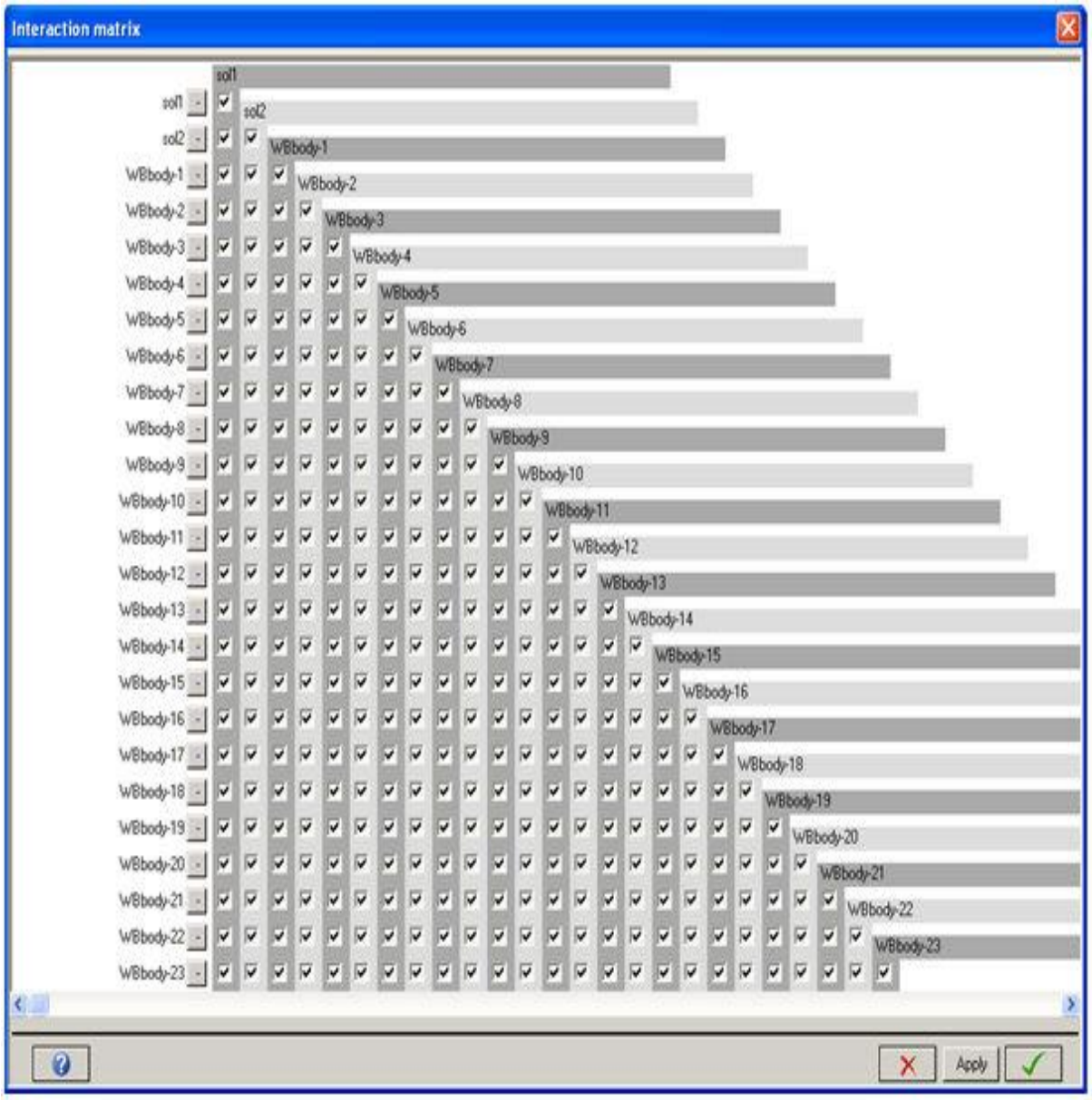
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This box also lists all Parts involved in interactions. Select one or more Parts in this list.

Press to remove interaction between all Parts selected in the top window and all Parts selected in the lower window.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Lagrange / Lagrange - Matrix



When the model doesn't contain more than 25 parts, this window lets you choose which Parts you want to interact by checking boxes in a matrix.

When a model contains more than 25 Parts the matrix option is not available and will be greyed-out.

If you have a large number of Parts, you may find it easy to set interactions using the **Add** / **Remove** options in the **Interactions** dialog panel.

The matrix shown in this panel list the Parts of your model vertically and horizontally. This matrix is symmetrical, so only the diagonal and lower part of the matrix is shown.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check a box if you want the two associated Parts to interact.

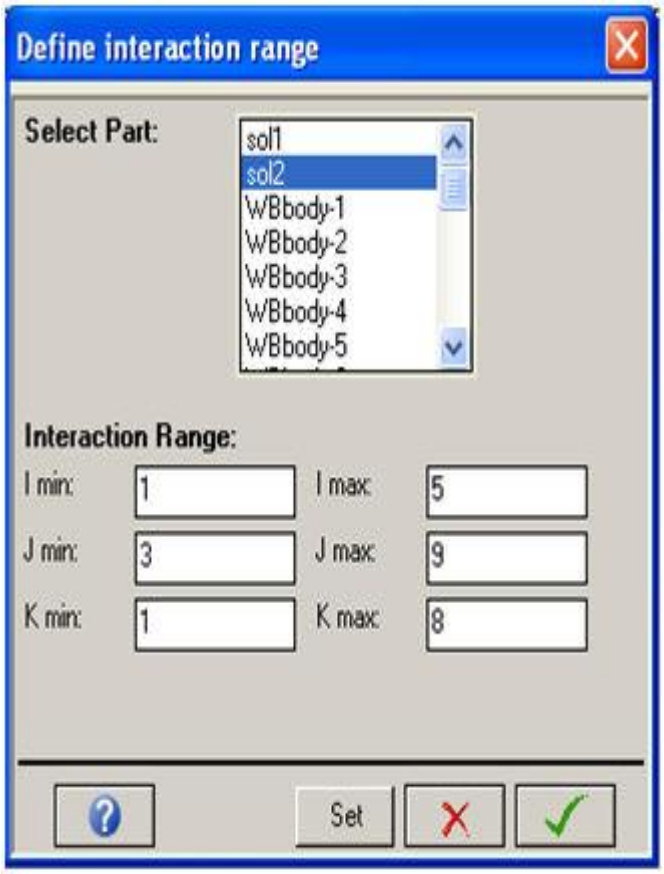
The Remove buttons let you quickly uncheck the boxes for each row of the matrix.

Press

to apply the changes you make to the matrix.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Lagrange / Lagrange - Range



This window is only available for Gap contact and lets you set the index range of a structured Part that will be checked for interactions. For unstructured parts this option is not available.

For Trajectory contact and for unstructured parts the Interaction by Group option can be used instead to only select a region of a Part for interaction.

Select Part

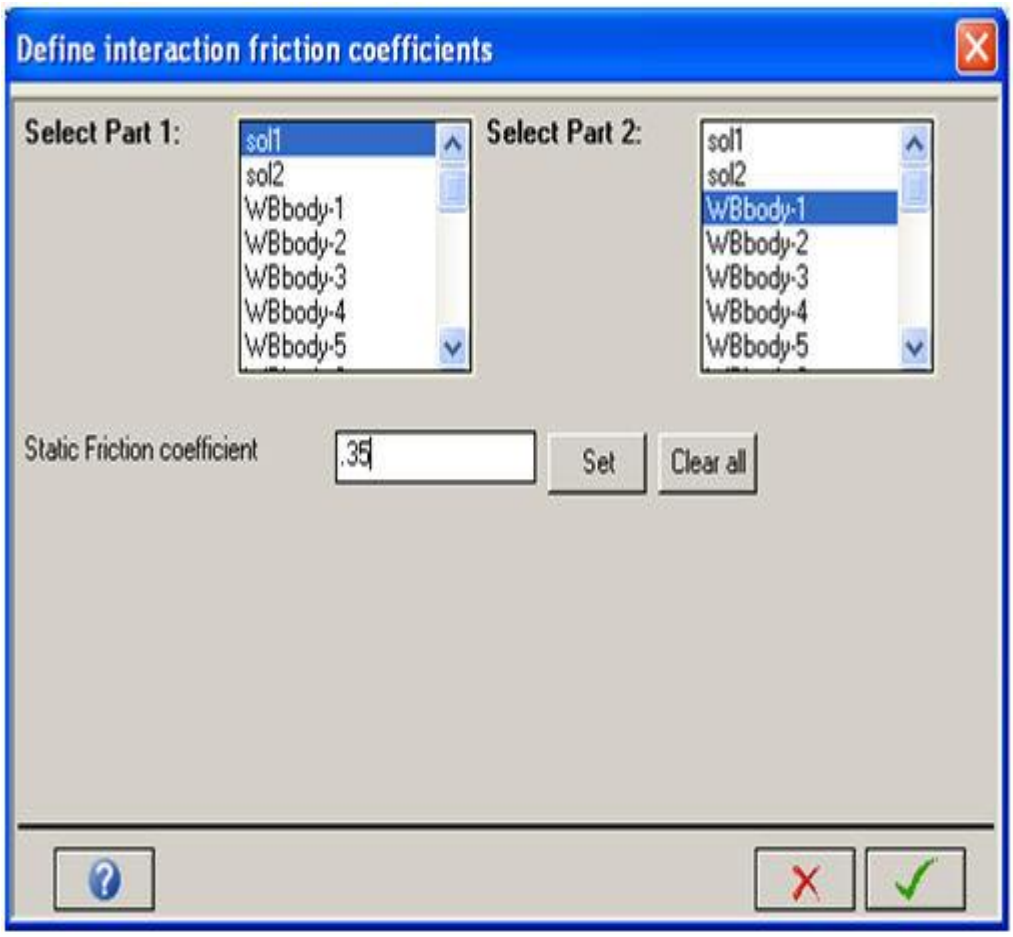
Select the Part for which you want to set the index range for interactions in this box.

Interaction Range

Enter the index range that you want to be checked for interaction.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Lagrange / Lagrange - Friction



This window is available for Gap contact and lets you set static friction coefficients between two Parts.

Select Part 1

Select the first of the two Parts you want to set the friction coefficient between in this box.

Select Part 2

Select the second of the two Parts you want to set the friction coefficient between in this box.

Friction Coefficient

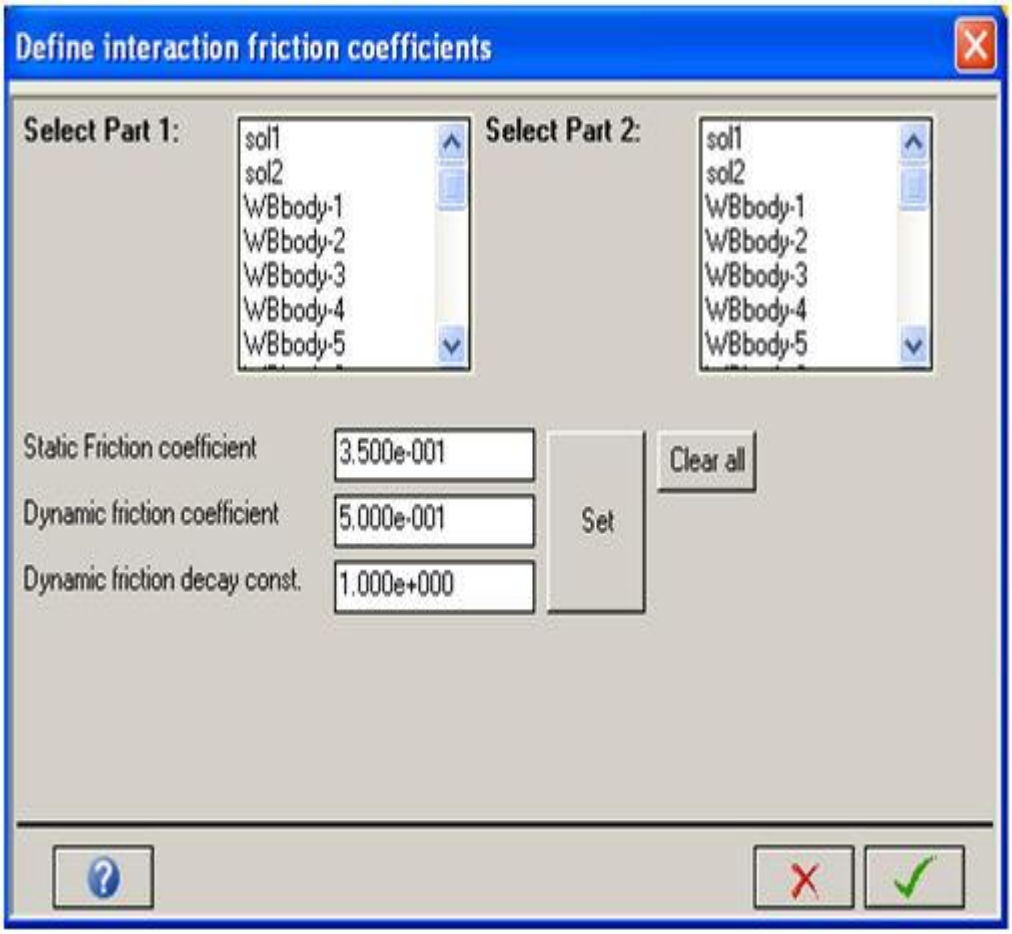
Enter the friction coefficient you want to use.

Press

to set the friction coefficient between the two selected Parts.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

You can apply this procedure as many times as you like to set coefficients between other Parts without exiting the window.



This window is available for Trajectory contact and lets you define static and dynamic friction between two Parts.

Select Part 1

Select the first of the two Parts you want to set the friction coefficient between in this box.

Select Part 2

Select the second of the two Parts you want to set the friction coefficient between in this box.

Friction Coefficient

A non-zero value for the Static Friction coefficient will activate Coulomb type friction between bodies (F=μR).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

A non-zero value for the Dynamic Friction coefficient and the Dynamic friction decay constant will activate the use of dynamic friction where an effective friction coefficient, μ, is used which is given by:

$$\mu = \mu_d + (\mu_s - \mu_d)e^{-\beta v}$$

where

μ_s = static
μ_d = dynamic coefficient of friction
β = exponential decay coefficient
v = relative sliding velocity at point of contact

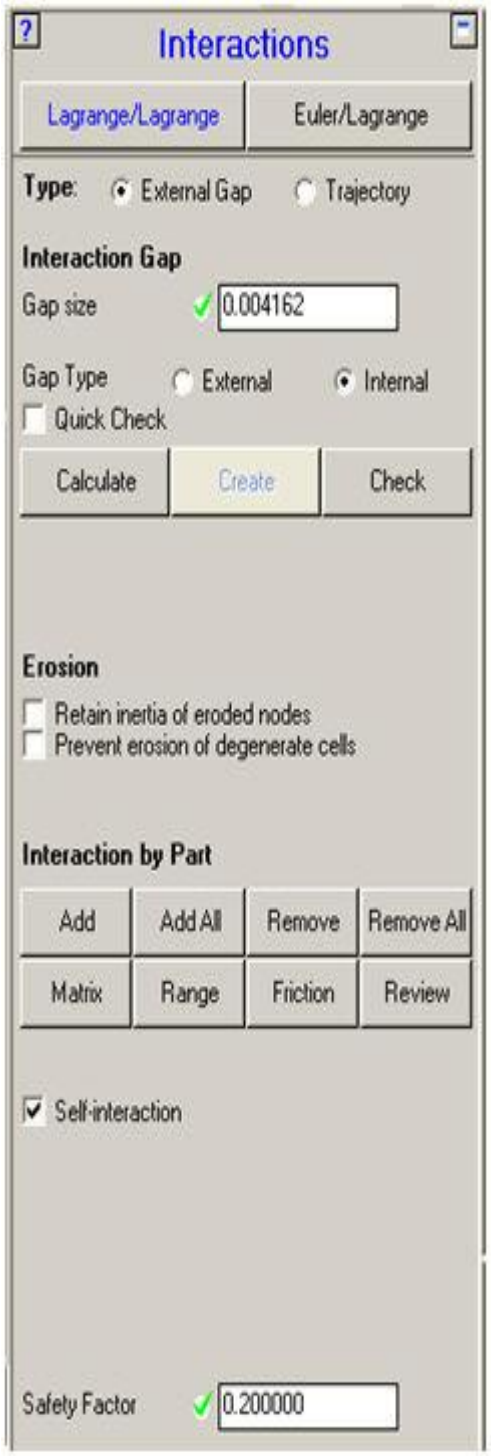
Press Set to set the friction coefficient between the two selected Parts.

You can apply this procedure as many times as you like to set coefficients between other Parts without exiting the window.

Note that friction is not available for Interaction defined by Group.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Lagrange/Lagrange - 2D Gap Contact Options



Type - External Gap

When a 2D model contains structured elements only, the 2D gap contact must be used to define Lagrange/Lagrange interactions.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

An analysis model that contains a mix of structured (IJK) Lagrangian Parts and unstructured Lagrangian Parts can currently not be run with Gap contact and also not with Trajectory contact.

To be able to use Trajectory contact the analysis model must contain unstructured Lagrangian volume Parts only. Similar, to be able to use Gap contact the analysis model must contain structured Lagrangian Parts only.

Interaction Gap

A gap is required for the interaction logic. The size of this gap must be in the range 1/10 to 1/2 the dimension of the smallest element face of Parts involved in interactions.

For most models you should set the gap size close to the smallest allowed value (Parts using a Lagrange or Shell Solver).

Gap size

Enter a value for the Gap size in this field or press **Calculate** to have AUTODYN set it for you.

Gap Type

Select whether you want to use an i**nternal** or **external** gap.

The external gap is **strongly** recommended because calculations using this option are more robust and more efficient. However, this option does require that interacting Parts are separated by the gap size at the start of your calculation.

Quick check

Check this box to use a fast calculation for the interaction time-step using only the maximum velocity in your problem. You should only check this box if you know that your maximum velocities will always be in impact regions.

Calculate

Press this button to have AUTODYN automatically calculate the Gap size for you. The Gap size will be set to 1/10 the dimension of the smallest element face of Parts involved in interactions.

Check

Press this button to check that the Gap size is valid and that all Parts involved in interactions are separated by the Gap size.

Erosion

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Retain inertia of eroded nodes

Check this box if you want to retain the inertial of eroded nodes (otherwise eroded nodes are removed from the model).

Prevent erosion of degenerate elements

Check this box if you want the interaction calculation to be used to stop elements from becoming degenerate. This option only works for Lagrange volume elements.

Warning! Activating this option can lead to unphysical results, particularly if a large Gap size is used.

Interaction by Part

Add

Press this button to set interactions between Parts.

Add All

Press this button to set all Parts to interact with each other.

Remove

Press this button to remove interactions between Parts.

Remove All

Press this button to remove all interactions between Parts.

Matrix

Press this buttons to set interactions using a Part matrix.

Range

Press this button to set the index range of each Part that will be checked for interactions.

Friction

Press this button to set static friction coefficients between different Parts.

Review

Press this button to review all your interaction settings.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Self Interaction

If the self-interaction option is switched on the contact detection algorithm will also check for external nodes of a part contacting with faces of the same part in addition to other parts. This is the most robust contact setting since all possible external contacts should be detected.

Interaction Timestep

AUTODYN automatically calculates an Interaction Timestep based on the Gap size and the maximum velocity of the Parts involved in interactions.

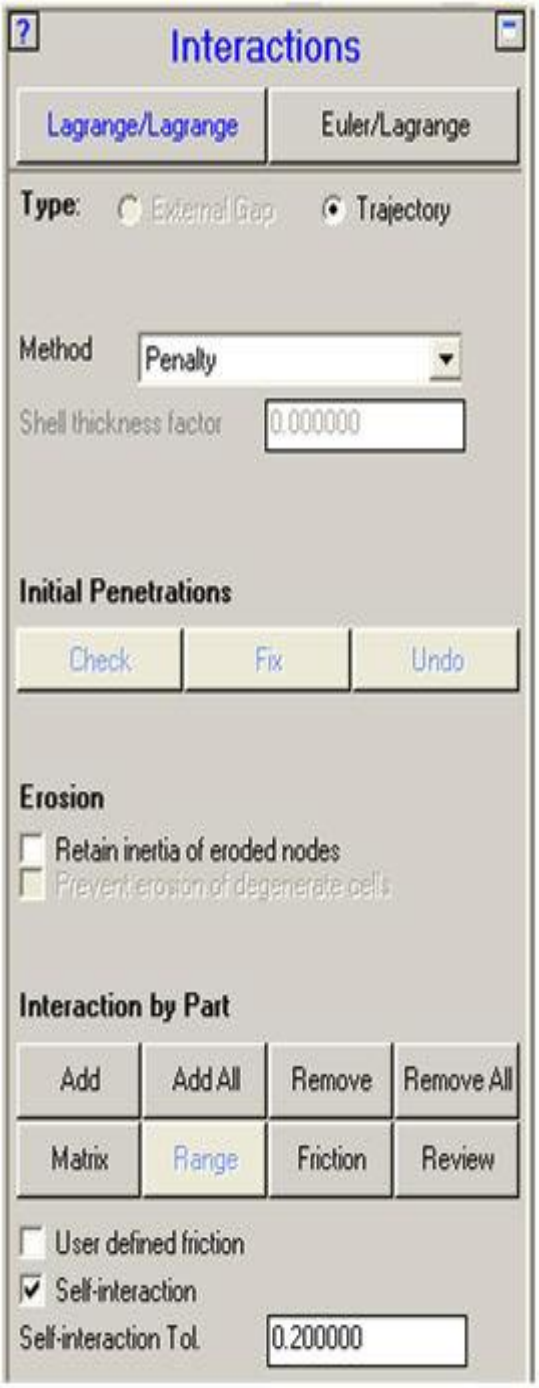
Safety Factor

Enter the safety factor you want to use to calculate the Interaction Timestep in this field.

You cannot enter a value greater than the default value (0.2).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions-Lagrange/Lagrange-2D Trajectory Contact Options



Type Trajectory

When a 2D model contains unstructured Lagrangian volume elements the 2D trajectory must be used to define Lagrange/Lagrange Interactions.

An analysis model that contains a mix of structured (IJK) Lagrangian Parts and unstructured Lagrangian Parts can currently not be run with Trajectory contact and also not with Gap contact.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

To be able to use Trajectory contact the analysis model must contain unstructured Lagrangian volume Parts only. Similar, to be able to use Gap contact the analysis model must contain structured Lagrangian Parts only.

Method

Method in which a contacting node is pushed back to the true contact position during the computational cycle.

There are two methods available:

1. **None**
   No contact defined

2. **Penalty**
   If a contact event is detected, a local penalty force is calculated to push the node back to the face. Equal and opposite forces are calculated on the nodes of the face in order to conserve linear and angular momentum. The applied penalty force will push the node back towards the true contact position during the cycle. However, it will take several cycles to satisfy the contact condition. Kinetic energy is not necessarily conserved. The conservation of energy can be tracked using the energy time history.

3. **Decomposition Response (DCR)**
   All contacts that take place at the same point in time are first detected. The response of the system to these contact events is then calculated to conserve momentum and energy. During this process forces are calculated to ensure that the resulting position of the nodes and faces does not result in further penetration at that point in time. The decomposition response algorithm is more impulsive (in a given computational cycle) than the penalty method. This can give rise to large hourglass energies and energy errors.

Retain inertia of eroded nodes

Check this box if you want to retain the inertial of eroded nodes (otherwise eroded nodes are removed from the model).

Interaction by Part

Add

Press this button to set interactions between Parts.

Add All

Press this button to set all Parts to interact with each other.

Remove

Press this button to remove interactions between Parts.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Remove All

Press this button to remove all interactions between Parts.

Matrix

Press this button to set interactions using a Part matrix.

Friction

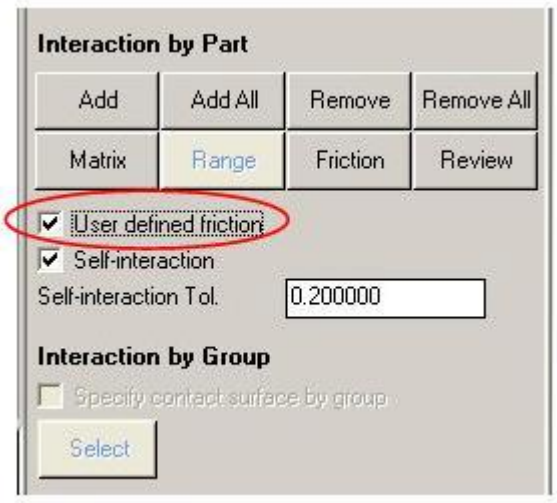
Press this button to define static and dynamic friction between different Parts.

Review

Press this button to review all your interaction settings.

User defined friction

Aside from the friction definition through the Interaction by Part method described above there is also a user defined friction option available through the use of a user-subroutine called EXFRICTION2D.



Self Interaction

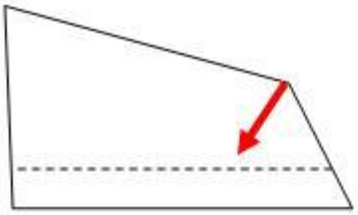
If the self-interaction option is switched on the contact detection algorithm will also check for external nodes of a part contacting with faces of the same part in addition to other parts. This is the most robust contact setting since all possible external contacts should be detected.

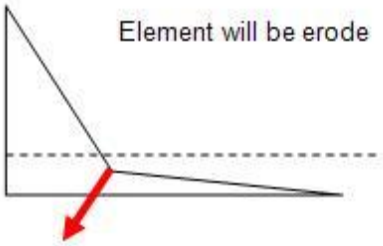
Self-interaction Tol

The self-interaction option enables automatic erosion when an element deforms such that one of its nodes comes within a specified distance of one of its faces. This option will prevent that volume elements become degenerated.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

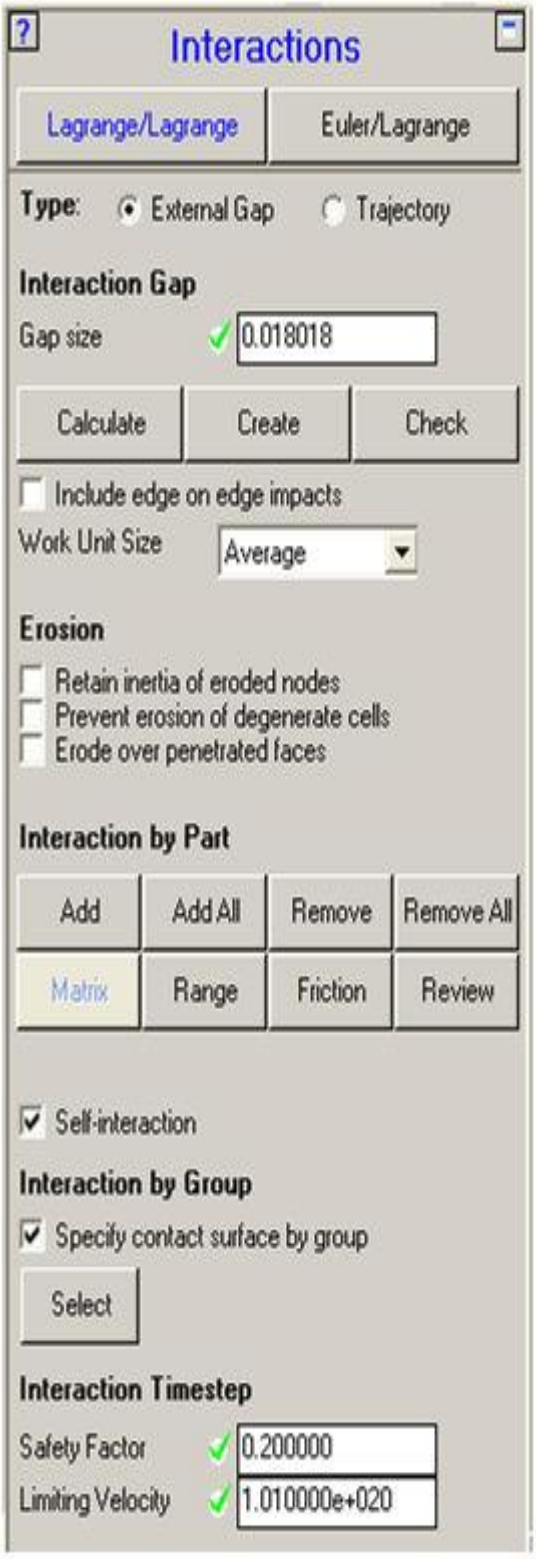
The specified distance is calculated using the Self-interaction Tol. value which is a factor in the range 0.1 to 05. This factor is multiplied by the smallest characteristic dimension of the elements in the mesh to give a physical dimension.





Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions-Lagrange/Lagrange-3D Gap Contact Options



Interaction Gap (External Gap options)

The External Gap contact algorithm is implemented for all Lagrange, ALE, shell, beam and unstructured solvers.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

A gap is required for the interaction logic. The size of this gap must be in the range 1/10 to 1/2 the dimension of the smallest element face of Parts involved in interactions.

There are two contact algorithms available in 3D, External Gap and Trajectory

For most models you should set the gap size close to the smallest allowed value (1/10 the dimension of the smallest element face).

Gap size

Enter a value for the Gap size in this field or press **Calculate** to have AUTODYN set it for you.

Calculate

Press this button to have AUTODYN automatically calculate the Gap size for you. The Gap size will be set to 1/10 the dimension of the smallest element face of Parts involved in interactions.

Create

All interacting Parts in you model must be separated by at least the Gap size. Press this button to automatically insert the required gap between Parts.

You may want to insert gaps manually using the **Translate** and / or **Scale** options in the zoning Panel Zoning panel.

Check

Press this button to check that the Gap size is valid and that all Parts involved in interactions are separated by the Gap size.

Include edge on edge impacts

Check this box if you want the interaction calculation to include testing for edge on edge impacts (significantly more computing is required if you check this box).

Work unit size
Select the size of work unit to be used for interactions.
Average - The work unit size is based on the average cell dimension
Smallest - The work unit size is based on the smallest element dimension
Largest - The work unit size is based on the largest element dimension

Erosion

Retain inertia of eroded nodes

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box if you want to retain the inertial of eroded nodes (otherwise eroded nodes are removed from the model).

Prevent erosion of degenerate elements

Check this box if you want the interaction calculation to be used to stop elements from becoming degenerate. This option only works for Lagrange volume elements.

Warning! Activating this option can lead to unphysical results, particularly if a large Gap size is used.

Erode over penetrated faces

Check this box if you want elements with over penetrated faces to be eroded.

You should only check this box if you know this option is required to successfully complete your calculation.

Interaction by Part

Add

Press this button to set interactions between Parts.

Add All

Press this button to set all Parts to interact with each other.

Remove

Press this button to remove interactions between Parts.

Remove All

Press this button to remove all interactions between Parts.

Matrix

Press this buttons to set interactions using a Part matrix.

Range

Press this button to set the index range of each Part that will be checked for interactions.

Friction

Press this button to set static friction coefficients between different Parts.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Review

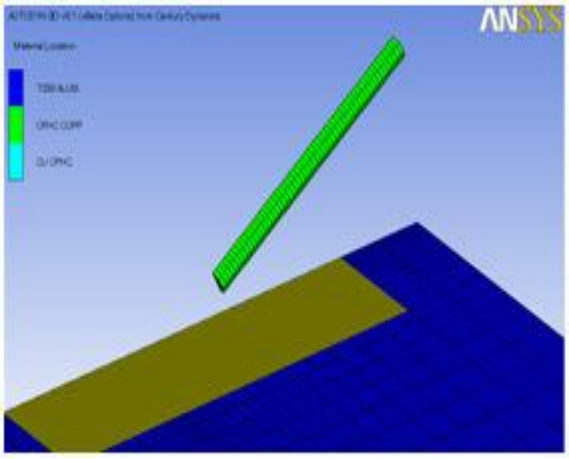
Press this button to review all your interaction settings.

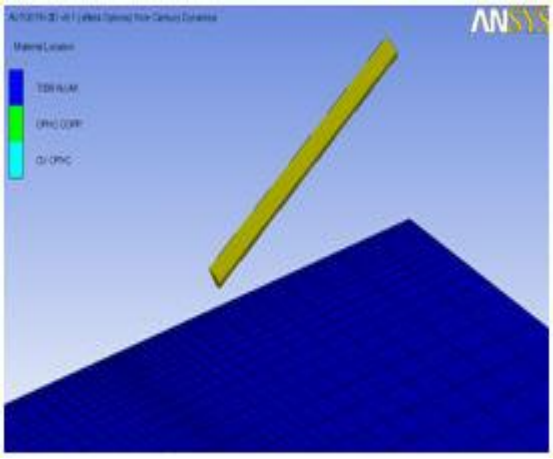
Self interaction of Parts

Check this box if you want Parts to interact with themselves (i.e. test for self-impact).

Interaction by Group

In many calculations, the contact region may be relatively small in comparison to the entire model, and as the interactions calculation is numerically intensive, it would be beneficial if the scope of its work were limited to this smaller area. The Group Contact option allows the user to select a set of face groups to describe the extent of the interactions in unstructured models.



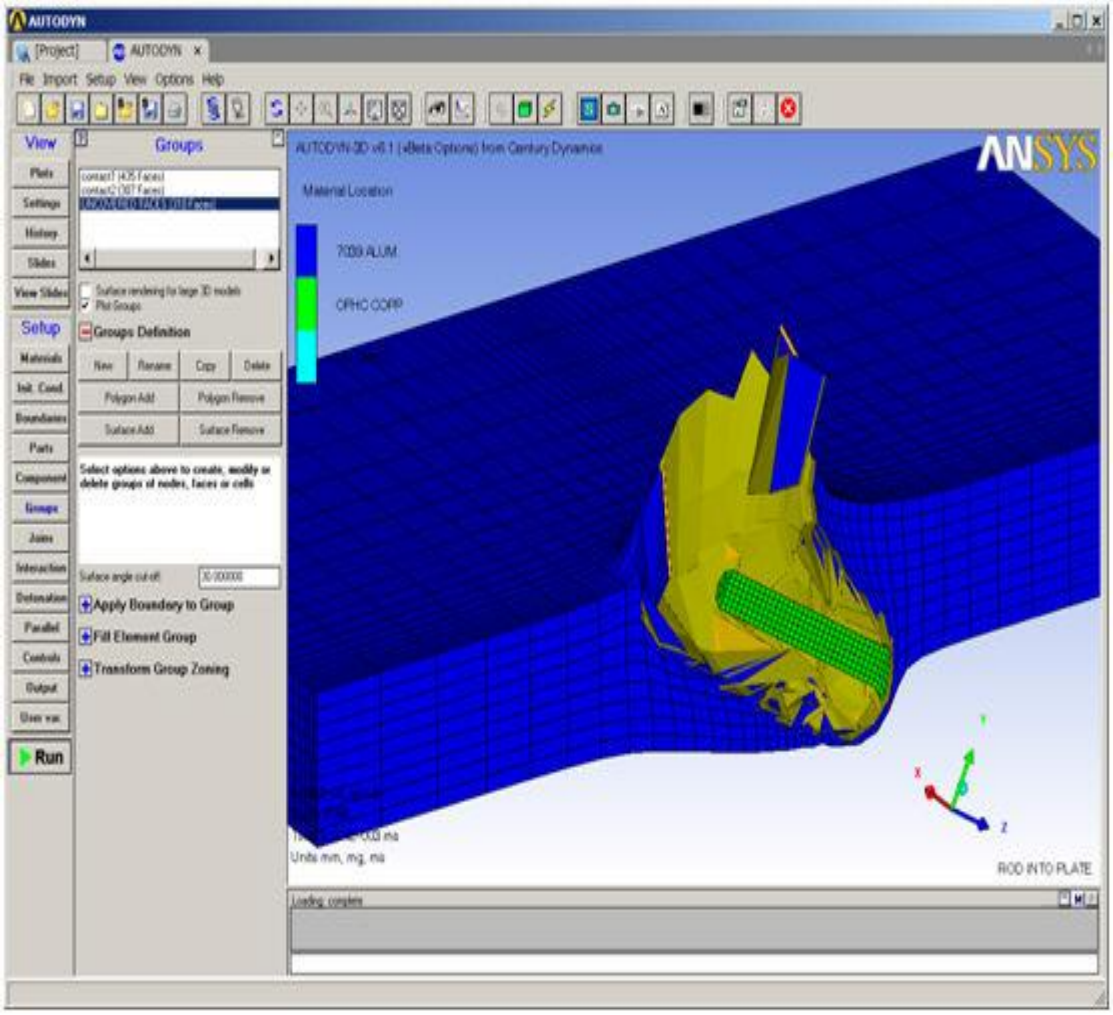


Select the "Specify Group Contacts by Group" toggle and press Select, to open the "Select Groups to include in Contact" dialog, and use the Add/Remove button to specify which face groups are to be included in the contact calculations.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

During execution, any faces eroded will be removed from the group, and any new faces uncovered will be added to a group named "Uncovered faces". Unstructured Beams and SPH can also be included as a node group to participate in contact.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Note that Group Contact is additive to Part contact. Selecting a Part for contact results in all external faces in the Part being checked for contact. Group contact should be used to add additional external faces from Parts that have been excluded from contact in the Part interaction matrix.

Group contact is only available to faces of unstructured Parts and doesn't include friction.

Interaction Timestep

AUTODYN automatically calculates an Interaction Timestep based on the Gap size and the maximum velocity of the Parts involved in interactions.

Safety Factor

Enter the safety factor you want to use to calculate the Interaction Timestep in this field.

You cannot enter a value greater than the default value (0.2).

Limiting Velocity

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the maximum velocity you want to use for calculating the Interaction Timestep
in this field.

This option is useful if you have large velocities controlling the timestep that are not involved in interactions.

The default limiting velocity is larger than any velocity your model should encounter.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions-Lagrange/Lagrange-3D Trajectory Contact Options

Interaction Trajectory (Trajectory options)

The Trajectory contact algorithm is implemented for all unstructured Lagrangian solvers and SPH.

Method

Method in which a contacting node is pushed back to the true contact position during the computational cycle.

There are three methods available:

1. **None**
   No contact defined

2. **Penalty**
   If a contact event is detected, a local penalty force is calculated to push the node back to the face. Equal and opposite forces are calculated on the nodes of the face in order to conserve linear and angular momentum. The applied penalty force will push the node back towards the true contact position during the cycle. However, it will take several cycles to satisfy the contact condition. Kinetic energy is not necessarily conserved. The conservation of energy can be tracked using the energy time history.

3. **Decomposition Response (DCR)**
   All contacts that take place at the same point in time are first detected. The response of the system to these contact events is then calculated to conserve momentum and energy. During this process, forces are calculated to ensure that the resulting position of the nodes and faces does not result in further penetration at that point in time. The decomposition response algorithm is more impulsive (in a given computational cycle) than the penalty method. This can give rise to large hourglass energies and energy errors.

Shell thickness factor

The value of the factor has to be taken between 0.0 and 1.0 and determines the amount of the shell thickness that is taken into account as the interaction distance. A factor of 1.0 takes the full shell thickness into account, which means that the contact surface is positioned at half the shell thickness on both sides of the shell mid plane.

A factor of 0.0 means that the shell has no contact thickness and the contact surface is positioned at the shell mid plane.

Note that for shell node on shell face impacts, the node is always located at the mid-surface of the shell.

Initial Penetrations

Check

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

On clicking this button, AUTODYN will search the model for initial penetrations of nodes into surfaces. If any penetrating nodes/surfaces are found, a new Group will be created so that they can be identified/displayed using the Groups panel. Nodes that initially penetrate surfaces will be "missed" by the contact algorithm.

Fix

When this option is selected the initially penetrated nodes are set back to the closest contact surface and no initial penetrations are present at the start of the analysis.

This option is meant for situations where small initial penetrations occur (for example because of round-off) and should be used with care because the analysis model geometry is changed.

Note that it may not be possible to remove all initial penetrations with this algorithm. Features are available in the DesignModeler and Meshing application to prevent initial penetrations occurring.

Undo

This option will restore the geometry of the analysis model before the initial penetrations were fixed.

It is recommended that the fixed initial penetrations are checked before the model is modified further otherwise it may not be possible to undo the last fix operation.

Erosion

Retain inertia of eroded nodes

Check this box if you want to retain the inertial of eroded nodes (otherwise eroded nodes are removed from the model).

Interaction by Part

Add

Press this button to set interaction between Parts.

Add All

Press this button to set all Parts to interact with each other.

Remove

Press this button to remove interaction between Parts.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Remove All

Press this button to remove all interactions between Parts.

Matrix

Press this button to set interactions using a Part matrix.

Range

Press this buttons to set the index range of each Part that will be checked for interactions.

Friction

Press this button to define static and dynamic friction between different Parts.

Review

Press this button to review all your interaction settings.

User defined friction

Aside from the friction definition through the interaction by Part method described above there is also a user defined friction option available through the use of a user-subroutine called EXFRICTION.

Self Interaction

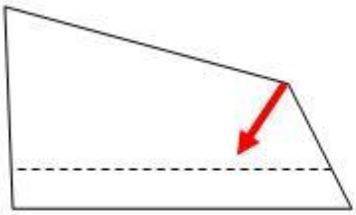
If the self-interaction option is switched on the contact detection algorithm will also check for external nodes of a part contacting with faces of the same part in addition to other parts. This is the most robust contact setting since all possible external contacts should be detected.

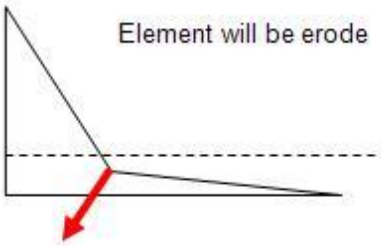
Self-interaction Tol.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The self-interaction option enables automatic erosion when an element deforms such that one of its nodes comes within a specified distance of one of its faces. This option will prevent volume elements becoming degenerate.

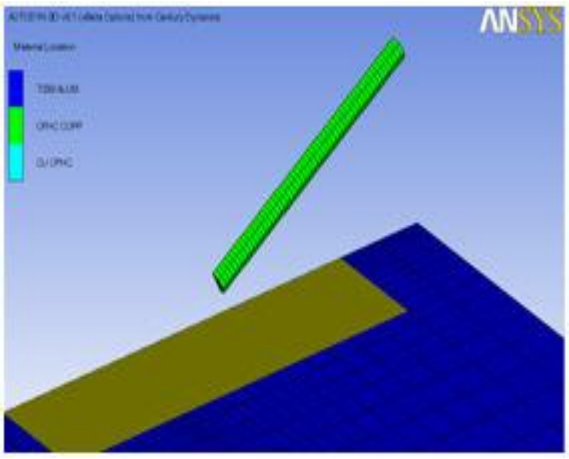
The specified distance is calculated using the Self-interaction Tol. value which is a factor in the range 0.1 to 0.5. This factor is multiplied by the smallest characteristic dimension of the elements in the mesh to give a physical dimension.

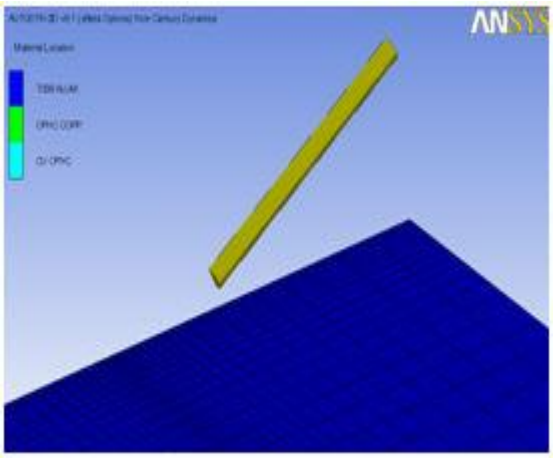




Interaction by Group

In many calculations, the contact region may be relatively small in comparison to the entire model, and as the interactions calculation is numerically intensive, it would be beneficial if the scope of its work were limited to this smaller area. The Group Contact option allows the user to select a set of face groups to describe the extent of the interactions in unstructured models.



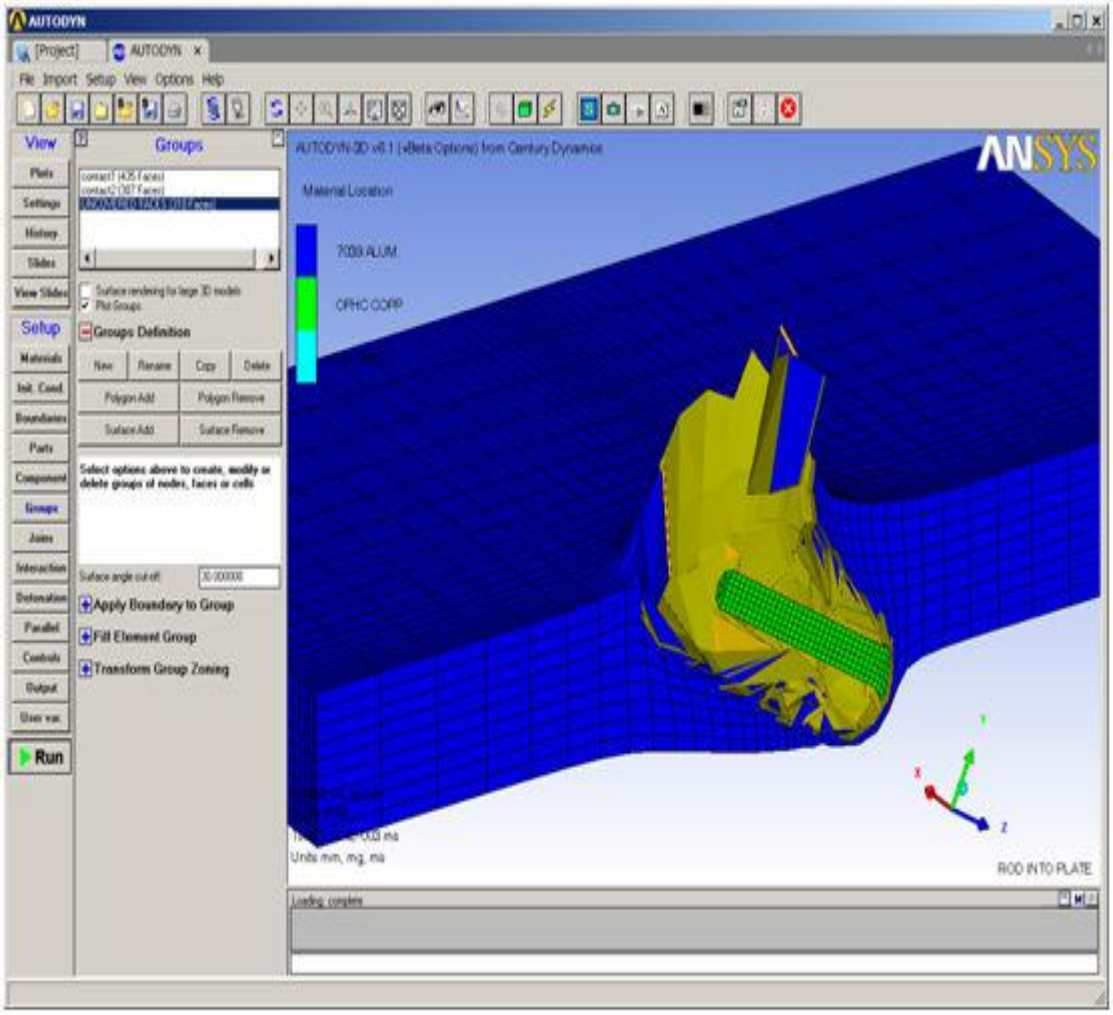


Select the "Specify Group Contacts by Group" toggle and press Select, to open the "Select Groups to include in Contact" dialog, and use the Add/Remove button to specify which face groups are to be included in the contact calculations.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

During execution, any faces eroded will be removed from the group, and any new faces uncovered will be added to a group named "Uncovered faces". Unstructured Beams and SPH can also be included as a node group to participate in contact.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



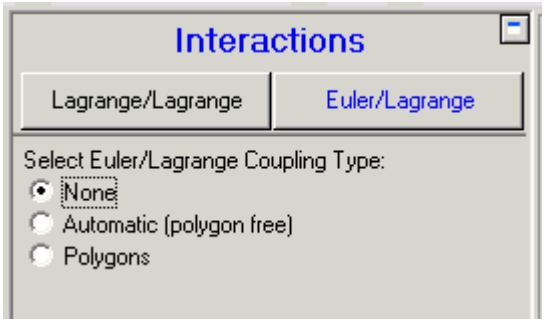
Note that Group Contact is additive to Part contact. Selecting a Part for contact results in all external faces in that Part being checked for contact. Group contact should be used to add additional external faces from Parts that have been excluded from contact in the Part interaction matrix.

Group contact is only applicable to faces of unstructured Parts and doesn't include friction.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Euler/Lagrange (2D)

Interactions - Euler / Lagrange (2D)



This panel lets you set up interactions (coupling) between 2D Lagrange Parts and Euler Parts.

Select Euler/Lagrange Coupling Type

Select the type of coupling you want to muse for Euler/Lagrange interactions.

None Do not compute any Euler/Lagrange coupling

Automatic Let AUTODYN set up coupling automatically.

Polygons Set up interactions manually using Polygons.

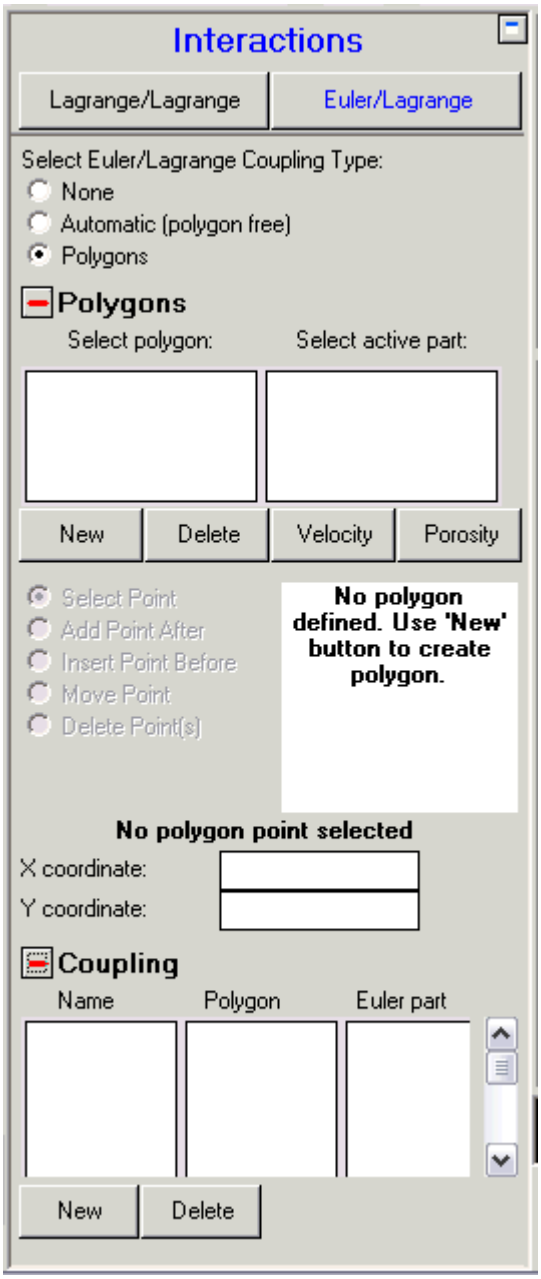
Automatic coupling is recommended if you want all Lagrange Parts to interact with all Euler Parts and you have no Shell Parts.

If you want more control in defining which Parts should be coupled or have Shell Parts that you want to couple to Euler, use **Polygon coupling.**

Erosion can be used with **automatic coupling**, but not with **Polygon coupling**.

If you select **Polygon coupling**, further input dialog is presented :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



To set up an Euler/Lagrange interaction using Polygons, you must first generate one or more Polygons and attach the polygon points to nodes on the surface of the Lagrange Parts that you want to interact with Euler.

Then you must specify which Polygons you want to couple with which Euler Parts.

This dialog lets you generate your Polygons and set up coupling between your Polygons and the Euler Parts in your model.

Polygons

Select Polygon

This box lists all the Polygons you have created. You can select a Polygon in this list.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select active part

This box lists all the Parts that can be coupled to Polygons (i.e. all Lagrange, ALE and Shell Parts). You can select a Part in this list.

New

Press this button to create a new Polygon.

Delete

Press this button to delete the selected Polygon

Velocity

Press this button to apply a velocity to the selected node.

Porosity

Press this button to set porosity for the selected Polygon.

Add First Point (only visible when no points are defined)

Use this option to generate the first point for the selected Polygon (use **shift + right mouse button** to place the point)

Select Point

Use this option to select a point of the selected Polygon (use **shift + right mouse button** to select the point)

Add Point After

Use this option to add a point after the selected polygon point (use **shift + right mouse button** to place the point)

If you place the added point close to a node on the surface of the selected active Part, the point will automatically be attached to that node.

If you immediately follow this by placing a second point close to another surface node of the same selected active Part, multiple points will be generated attached to all surface nodes between the two placed nodes in an anti-clockwise direction.

Insert Point Before

Use this option to insert a point before the selected polygon point (use **shift + right mouse button** to place the point)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

If you place the inserted point close to a node on the surface of the selected active Part, the point will automatically be attached to that node.

If you immediately follow this by placing a second point close to another surface node of the same selected active Part, multiple points will be generated attached to all surface nodes between the two placed nodes in an anti-clockwise direction.

Move Point

Use this option to move the selected polygon point to a new location (use **shift + right mouse button** to place the point at its new location)

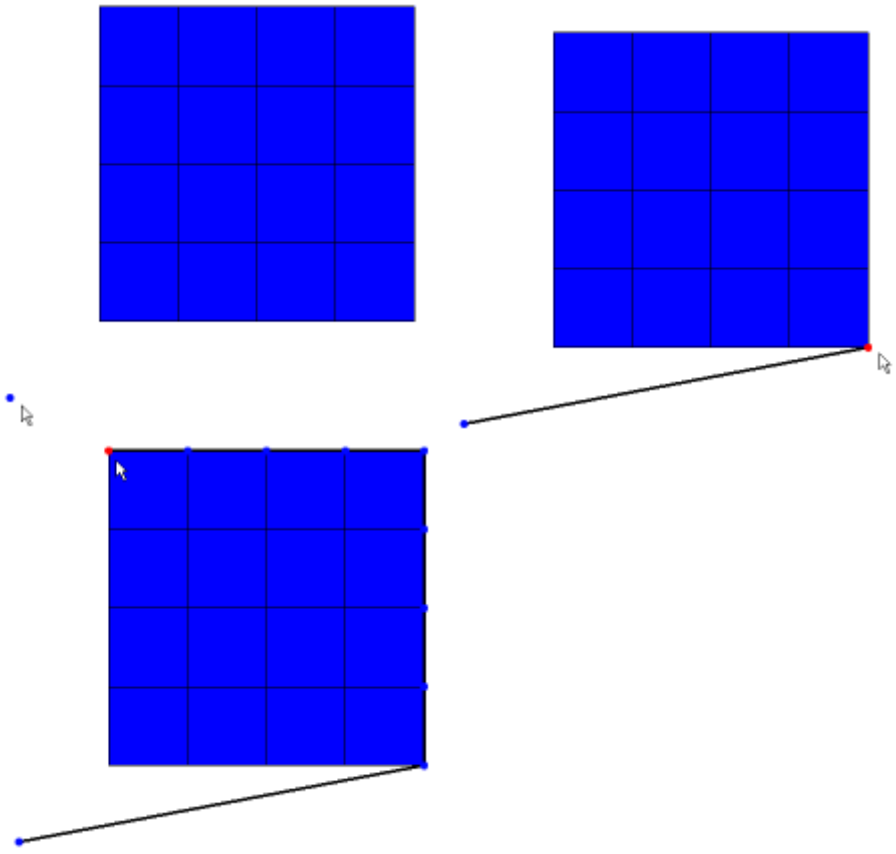
Delete Point(s)

Use this option to delete one or more polygon points. The first point to be deleted is the selected polygon point. Use **shift + right mouse button** to select the last point to be deleted. All points from the selected point to last point inclusive will be deleted.

X/Y Coordinates

These fields display the coordinates of the selected polygon point. You can change the location of this point by entering new values in these fields.

Example



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Add first point
Add point after
Add point after
(Single point not attached to grid)
(Single point attached to grid)
(Multiple points generated

anti-clockwise around grid)

The selected polygon node is always shown in red.
The last point of Polygon is automatically connected to first point to close the Polygon.

Coupling

Name / Polygon / Euler Part

These boxes list the interaction sets you have defined for Polygon coupling.

Name The name of the interaction set.
Polygon The Polygon that is coupled to an Euler Part.
Euler Part The Euler Part that the Polygon is coupled to.

You can select a set in this list

New

Press this button to add a new interaction set.

Delete

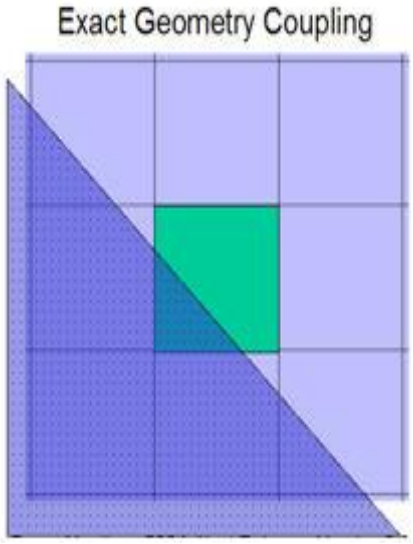
Press this button to delete the selected interaction set.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler / Lagrange Introduction

The Euler/Lagrange interaction (Fluid-Structure Interaction) in AUTODYN couples the Eulerian domain to the Lagrangian structural domain.

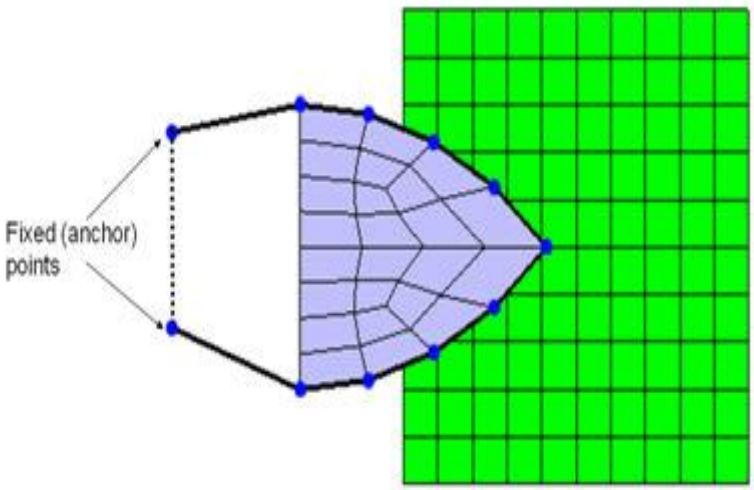
This coupling algorithm in AUTODYN finds the intersection between the external faces of the interacting structures and the cells and faces of the Euler grids. These intersection calculations give an exact calculation (within numerical precision) of the covered fraction of the Euler cells and faces, as shown in the left hand figure below. These properties are used in the 3D Euler solvers for coupled simulations.



The coupling can be chosen to be fully automatic (2D/3D) or by means of polygons (2D).

Polygon coupling
This coupling technique is available in 2D to set up an Euler/Lagrange interaction where one or more Polygons are defined and the polygon points are manually attached to nodes on the surface of the Lagrange Parts that interact with Euler. All structured Lagrangian element types can be coupled to Euler using polygons. Unstructured parts cannot be coupled to Euler using polygons; in that case the automatic coupling needs to be used.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

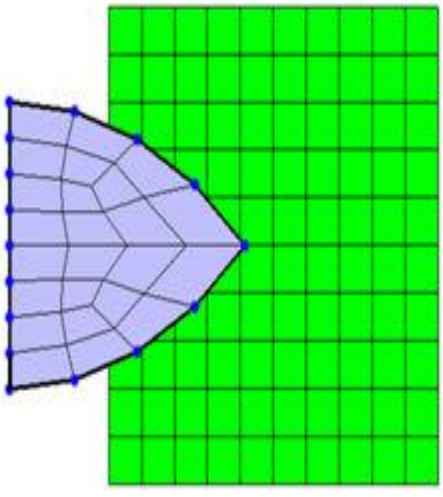


Automatic coupling

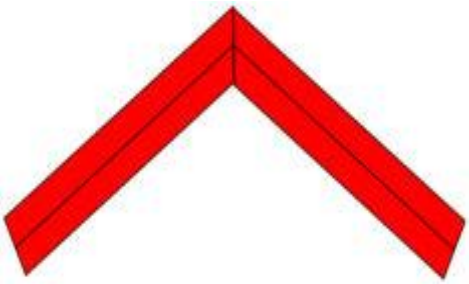
This coupling technique is available in 2D as well as in 3D. AUTODYN will automatically setup the Euler/Lagrange interactions and the only thing the user has to define is which parts interact with Euler. Automatically all outside surface segments of all selected Lagrangian parts are generated and coupled to Euler.

In 2D the automatic coupling is only available for structured and unstructured Lagrangian volume elements.

In 3D the automatic coupling is available for structured and unstructured Lagrangian volume and shell elements. For shell elements an artificial thickening of the shell parts is used.



Automatic generation of outside surface



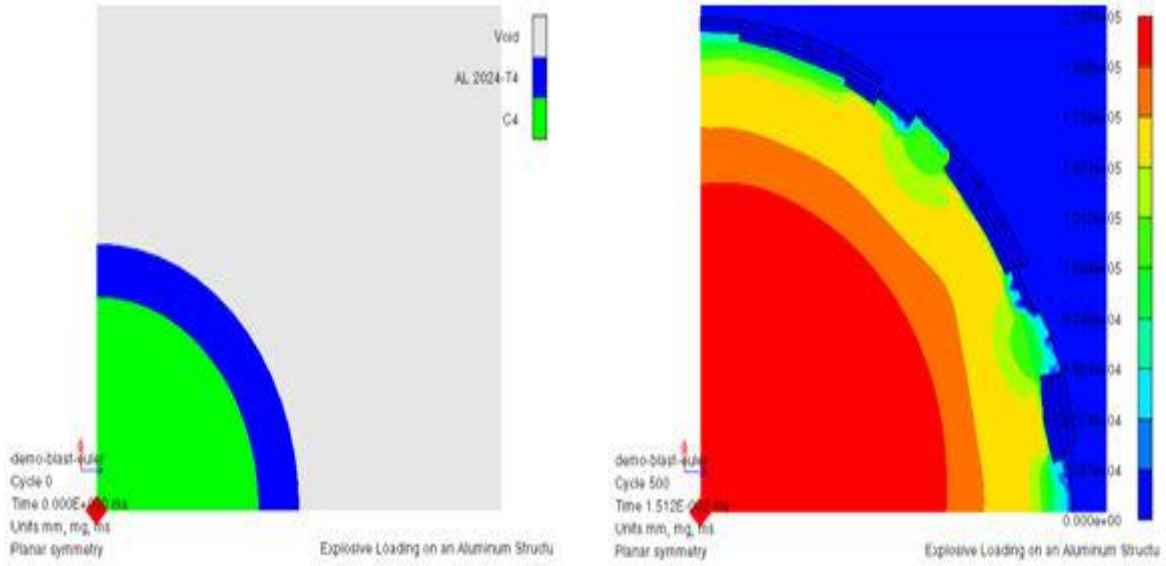
Artificial shell thickening for coupling

Coupling and Erosion

Erosion in combination with Euler/Lagrange coupling can be used for automatic coupling in 2D as well as 3D.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

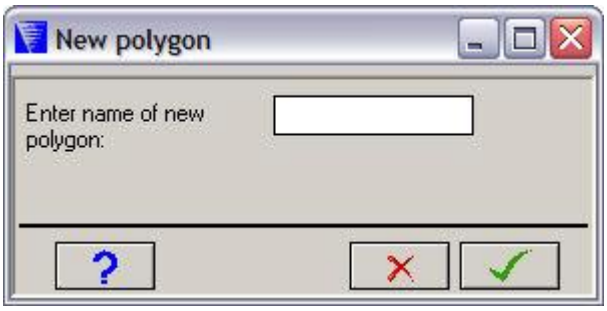
When elements that are connected to the coupling surface are eroded the coupling surface automatically will be updated for the new uncovered Lagrangian element faces.



Example simulation of fragmenting ring using Euler-Lagrange Coupling and requiring automated update of coupling surfaces during the simulation

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler/Lagrange (2D) - New Polygon



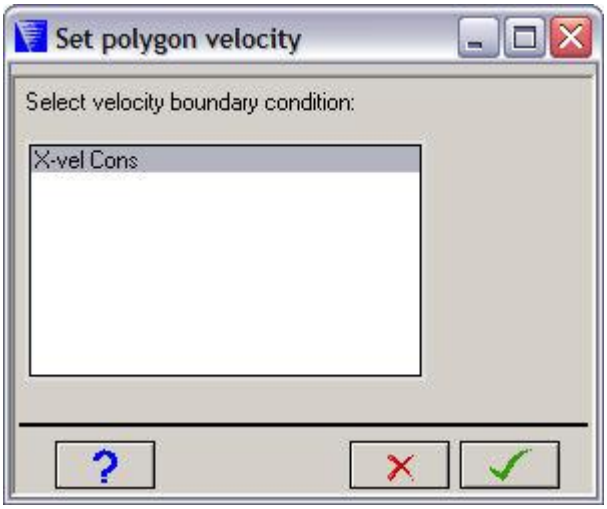
This window lets you create a new Polygon for Euler/Lagrange coupling.

Enter the name you want to use for your new polygon in the field provided and press .

Once you have created your Polygon, you can add points and attached these points to Lagrange nodes using options in the Dialog panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler/Lagrange (2D) - Velocity



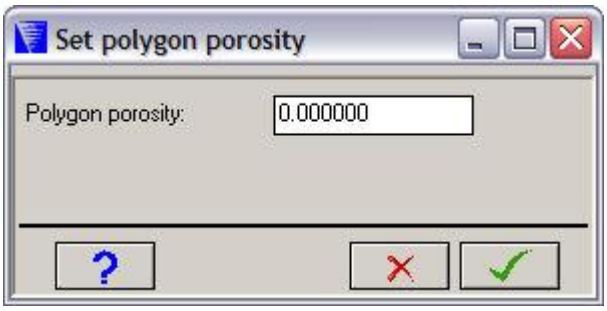
This window lets you attach a velocity boundary condition the selected polygon.

All polygon points not attached to a Part will use this velocity.

Select the boundary condition you want to use from the list and press .

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler/Lagrange (2D) - Porosity



This window lets you set porosity for your Polygon.

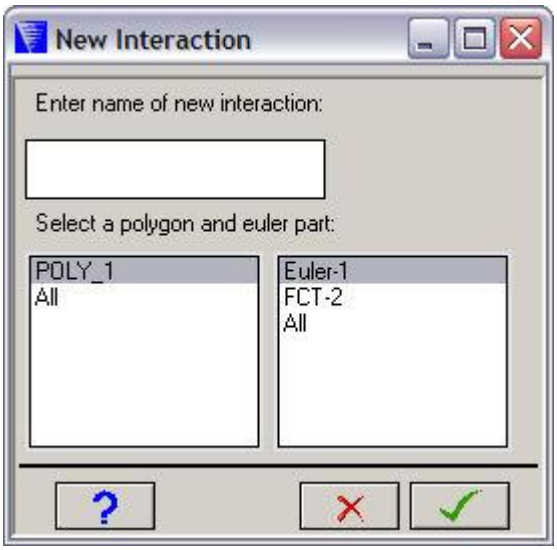
By default, a Polygon completely covers the Euler Parts it is coupled to.

Setting porosity greater than zero allows partial flow in the Euler Parts under the Polygon.

Enter a porosity factor between 0.0 and 1.0.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler/Lagrange (2D) - New Interaction



This window lets you define a new interaction set for Polygon coupling.

Enter name of new interaction

Enter the name you want to use for the new interaction set in this field.

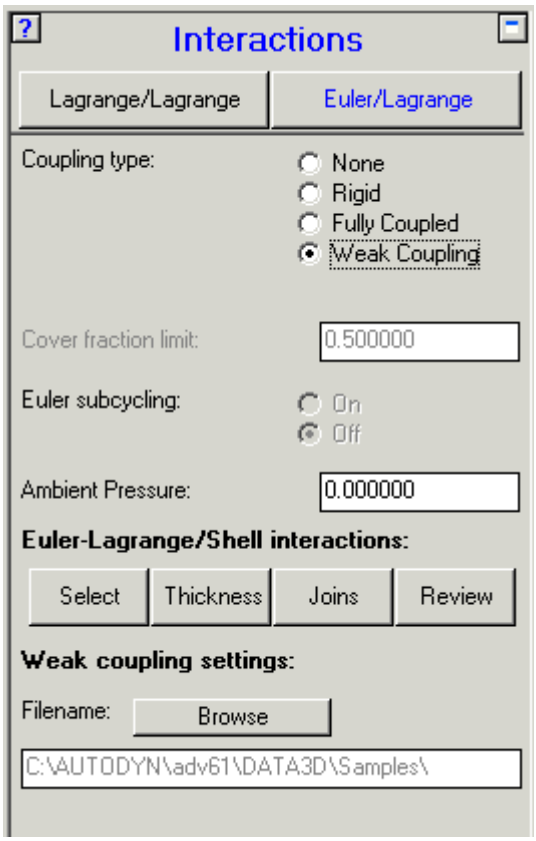
Select a Polygon and Euler Part.

Select the Polygon in the left box that you want to be coupled to the Euler Part you select in the right box.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Euler/Lagrange (3D)

Interactions - Euler - Lagrange (3D)



This panel lets you set up interactions (coupling) between 3D Lagrange Parts and Euler Parts.

Coupling Type

Select the coupling type you want to use.

None	- no coupling is defined
Rigid	- Fill Parts are used to define rigid boundaries for Euler.
Fully Coupled	- Lagrange Parts interact with Euler Parts dynamically.
Weak Coupling	- A fast method for having Lagrange and Euler Parts interact dynamically.

Cover fraction limit

Enter the cover fraction limit in this field. This is used to determine when a partially covered Euler cell is blended to a neighbor.

Euler subcycling

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Select whether you want Euler subcycling **On** or **Off**

Euler/shell interactions

Select

Press this button to select the Lagrangian Parts you want to couple to Euler.

Thickness

Press this button to specify the shell thickness you want to use for the coupling calculation.

Joins

Press this button to specify which shells are to be joined for the coupling calculation.

Review

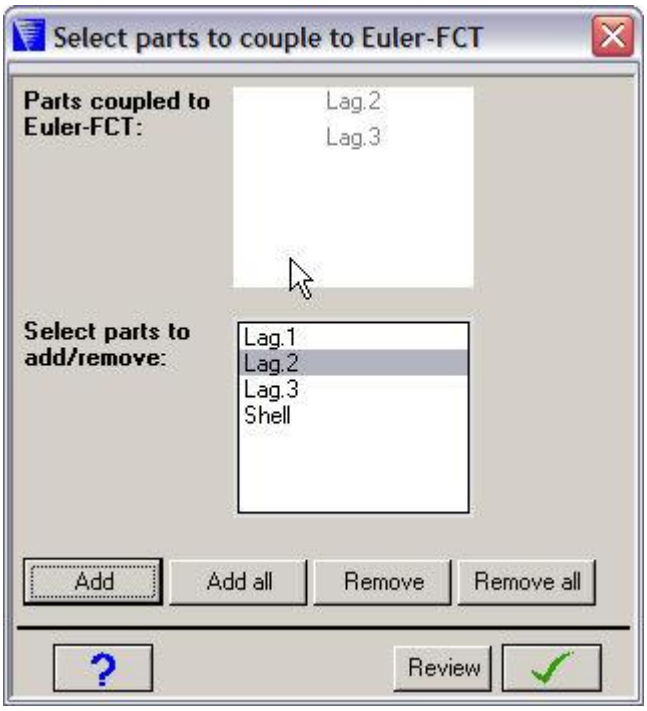
Press this button to review your Euler-Shell interactions.

Weak coupling settings

Browse for weak coupling data file.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler / Lagrange (3D) - Select



This window lets you select the Lagrangian Parts you want to couple to Euler.

Parts coupled to Euler-FCT

This box shows you the current Parts that are coupled to Euler.

Select parts to add/remove

This box shows all the Parts that can be coupled to Euler. You can select one or more Parts in this list.

Add

Press this button to add the selected Parts to be coupled with Euler.

Add all

Press this button to add all the Parts to be coupled with Euler.

Remove

Press this button to remove the selected Parts from being coupled with Euler.

Remove all

Press this button to remove all Parts from being coupled with Euler.

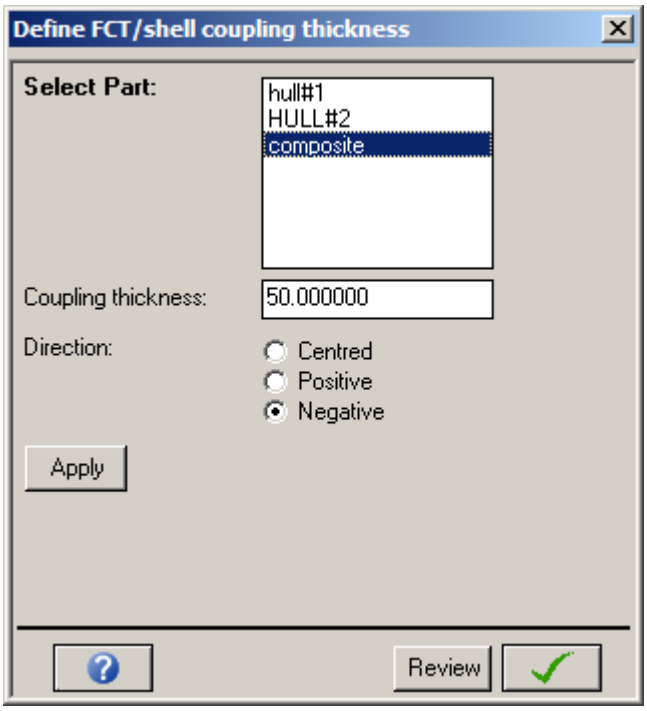
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Review

Press this button to add review all coupling with Euler.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler / Lagrange (3D) - Thickness



As well as being able to couple volume element solvers, it is also possible in AUTODYN to couple shell solvers. For a shell solver, the parts do not have any geometric through thickness dimension, and as such cannot cover any volume in the Euler mesh. Therefore, each shell part needs to be artificially thickened by the user.

Select Part

This list contains all the shell parts in the model that are selected for coupling with Euler. If you select a part, the coupling thickness and direction of the thickening for that part is shown below.

Coupling Thickness

This shows the coupling thickness for the selected part. To modify the coupling thickness, enter the new value, and press the Apply button. For the coupling methodology to function correctly, the artificial thickness of a shell must be at least twice the dimension of the largest cell size in the surrounding Euler grid.

Direction

This shows the direction the shell is thickened in for the selected part. To modify the thickening direction, select the desired direction, and press the Apply button.

The coupling thickness can be applied on both sides of the midplane of the shell (Centred) or applied to one side of the shell, where Positive is in the direction of the normal of the shell.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

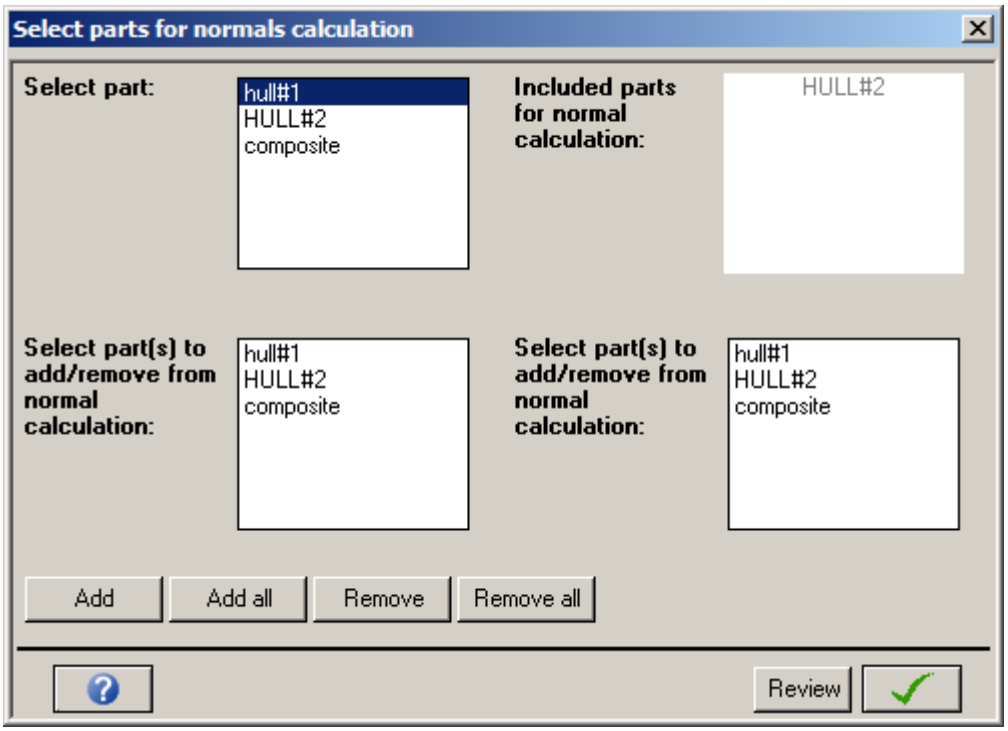
Review

If you select the review button, an html summary is opened that displays which shell parts have been included in the Euler coupling, the thickness and direction for these shell parts, and also the thickened shell joined matrix.

You can also view the thickened shells by switching on "Shell thickness" in the Plots panel, and choosing "Artificial Shell Coupling Thickness" in the Shell Thickness Settings panel.

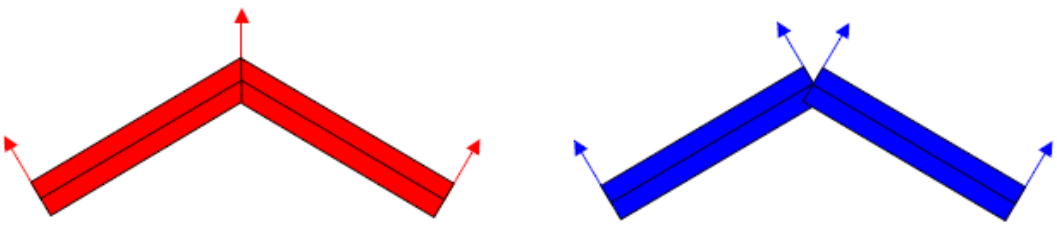
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler / Lagrange (3D) - Joins



To ensure that volumes of joined thickened shells do not overlap as the shells deform, the thickening algorithm needs some information on which shell parts should be considered joined when computing the outward normals for shell nodes in the thickening geometry calculations. By default all thickened shell parts are set as joined for the coupling calculation. If this is not appropriate, for example for a T-junction between two shell parts, you should remove the joins for the coupling calculation. It is important to note that these join settings are only used in the coupling calculation and do not affect the structural joins.

This is illustrated in the figure below. On the left, the two shell parts are joined for thickening calculations, and the resulting thickened shells define a closed volume. On the right, the two shell parts are not joined for thickening calculations, and the shell overlap is not satisfactory for coupling.



Select Part

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This list shows all the shell parts that are coupled in Euler. When a part is selected in this list, all parts joined to it are displayed in the "Included parts for normal calculation" list.

Included parts for normal calculation

This list shows all shell parts that are joined to the shell part selected in "Select Part"

Select part(s) to add/remove from normal calculation

These two fields are used to add or remove parts from the normal calculation. For example, a model has shell parts "hull#1", "hull#2" and "composite", as shown in the image of the window at the top of the page. To include the normals between the first two parts, select "hull#1" in the left hand list, "hull#2" in the right hand list, and press the apply button.

Review

If you select the review button, an html summary is opened that displays which shell parts have been included in the Euler coupling, the thickness and direction for these shell parts, and also the thickened shell joined matrix, that shows which shells are included in the normal calculation.

You can also view the thickened shells by switching on "Shell thickness" in the Plots panel, and choosing "Artificial Shell Coupling Thickness" in the Shell Thickness Settings panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Interactions - Euler / Lagrange (3D) - Weak Coupling

Introduction

Within AUTODYN-3D, there is a facility for the application of a pressure field in 3D space to Lagrange type parts as a boundary condition. The pressures in 3D space are contained in a sequence of *Field* files, the format of which is described below. Also included is an additional feature that allows the user to create these fields when using either the FCT or the Euler processors. Weak Lagrange Coupling can be seen as equivalent to the current FCT - Lagrange couple but without the feedback from the Lagrange type parts back into the FCT part. This weak coupling assumes that any deflections in the Lagrange model are small, and should only be used in such cases.

The current implementation only considers the possibility of mapping pressure data, and applies this to the Lagrange boundary surfaces as nodal forces. The pressures will be applied to all wetted surfaces of Lagrange type subgrids within the volume described by the pressure files. The advantage over the current LCT-Lagrange coupling is that multiple Lagrange calculations can be carried out on one FCT calculation, thus saving analysis time. Also, some speedup would be expected over a normal coupled solution for a single FCT and then Lagrange calculation.

Creating the Field files

The *Field* files are created as a field edit when an FCT or Euler type simulation is being carried out. The user should press **Output** on the navigation bar and open the **Field output** part of the dialog panel. Under this menu there are three options, namely ***Cycles***, ***Times*** and ***Set_part***. The cycle and time options work in the traditional manner of all edits in AUTODYN and allow the user to select the frequency of the Field file output. The third menu, set_part, allows the user to select which part the pressure data will be taken from, and the I, J and K range. The file format only allows one block of data to be written. If this information is not given by the user then the entire range of the first ECT or Euler part found will be written to the Field file. A geometry data field is only produced when a problem is started or restarted, and this cycle coincides with a field edit.

Using the Weak Interactions

The coupling option ***Weak Coupling*** is included under the Euler Lagrange interaction tab after ***Interaction*** is selected from the navigation bar. The Euler-Lagrange/Shell interactions need to be set to chose which Lagrange type part are included in the coupling, and how shells interact. Under the weak coupling settings, the ***Ambient Pressure*** should be entered. This pressure is removed from the Field file pressure, i.e.

$$P_{APPLIED} = P_{FIELD} - P_{AMBIENT}$$

Also the Field file set need to be selected by using the file browser to select the first in the required set.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Notes on the Mapping

AUTODYN-3D interpolates linearly in time the pressure data from two Field files, the current time of the simulation being in-between the times of the fields. When a field file is read into the solver a mapping is created. This is a lookup table for each cell face in the Lagrange model that is activated pointing to a cell in the pressure field. If a geometry data field is not present in the Field file then the existing mapping from the previous field is used. The mapping is always created using the initial coordinates (i.e. at time equal to zero) of the Lagrange type parts. The interpolation in space is the same as for the current FCT-Lagrange interactions; the nearest cells centre to the face centre is used.

Field file format (.ADF FILES)

The ADF (AUTODYN FIELD) file format contains field information on a regular structured rectangular grid. Scalar data is cell centred. The naming convention of the files is

IDENT_####.ADF FORMAT:'(A,A1,I4.4,A4)'

Where #### is an increasing integer number, starting at 0001.

Record 1	FILEID (CH*256)	Character string which can be used to identify the file
Record 2	NCYCLE, TIME, NTU, NMU, NLU	NTU is the time unit (1:μs 2:ms, 3:s) NMU is the mass unit (1:mg 2:g, 3:kg) NLU is the length unit (1:mm, 2:cm, 3:m)
Record 3	NI,NJ, NK	Maximum number of nodes in each of the three directions
Record 3+(NS-1)*2	ITEM_NAME (CH*256)	Name of the item to follow, i.e. GEOMETRY or PRESSURE
Record 4+(NS-1)*2	The data attached to the item	Item specific - See below
Record (LAST)	"END OF SCALAR FILE"	

The data attached to the items are

GEOMETRY

XC (1:NI)	X positions, inline with the I axis
YC (1:NJ)	Y positions, inline with the J axis
ZC (1:NK)	Z positions, inline with the K axis
PN(1:NI-1, 1:NJ-1, 1:NK-1)	Pressure of each cell. I varies fastest

A geometry data record is only needed if the geometry has changed. The presence of this field enforces a new mapping. Therefore, if the geometry is only present in

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

the first field, then the mapping is only done at the beginning of the calculation. On the other hand, if the geometry is present in every file, then a remapping is done every time a new flow file is read. If a file does contain a geometry data record then AUTODYN expects that it will occur before the pressure record. A pressure record must exist in all files.

Errors, Warning and other Outputs

Errors encountered during the mapping cause a generic error, stop the current simulation. A summary of the error is written to the message window. The following table summarizes the error codes.

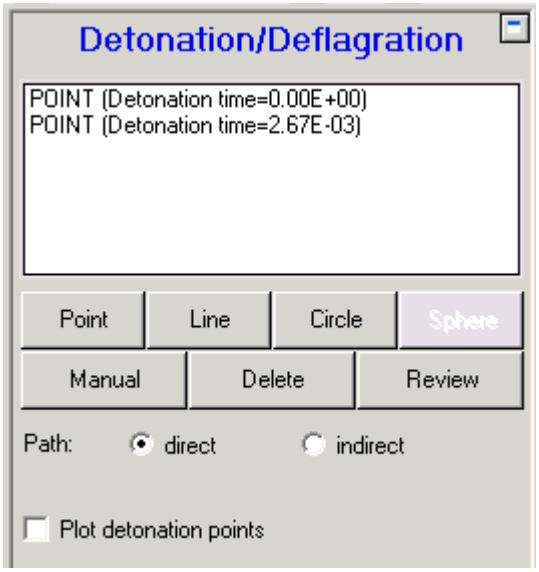
Too many scaler files	1
File does not exist	10000 + file counter
Expecting geometry data	20000 + file counter
No pressure data	30000 + file counter
Error in reading file	40000 + file counter

Errors and Warnings in the Field File writing routines will not stop the calculation, but will write to the message window an appropriate message.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (2D)

Detonation (2D)



This window lets you set the locations where explosives will be detonated / deflagrated.

Detonations / Deflagrations

The box at the top of the window shows you a list of the current detonations / deflagrations you have defined. You can select from this list.

Point

Press this button to define a detonation point.

Line

Press this button to define a line of detonation.

Circle

Press this button to define a detonation circle (planar symmetry).

Sphere

Press this button to define a detonation sphere (axial symmetry).

Manual

Press this button to manually define initiation times for blocks of elements.

Delete

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press this button to delete the selected detonation definition

Review

Press this button to review information for all detonation definitions.

Path

Select whether you want detonation times to be calculated using direct or indirect paths.

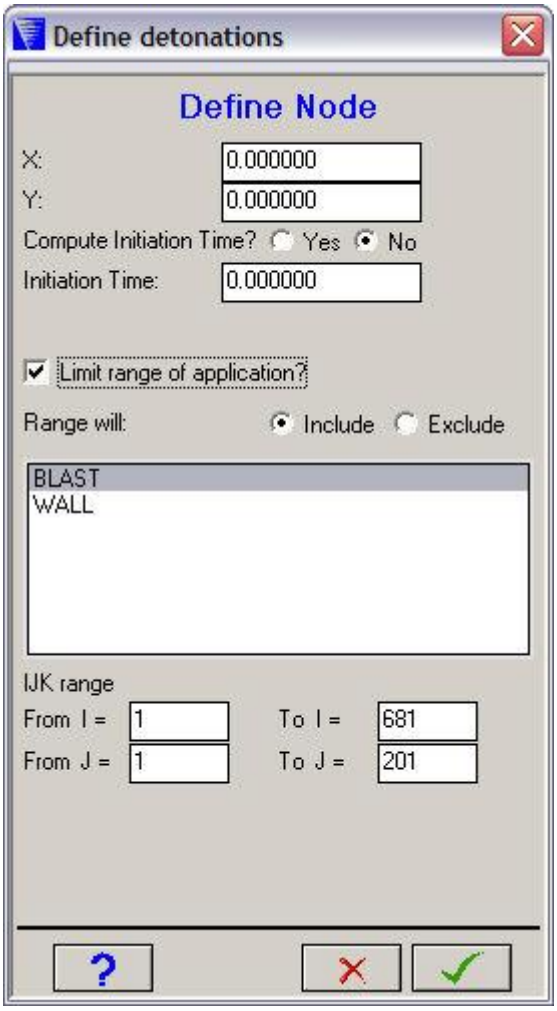
> **Direct Path** : Detonation times are computed from straight line paths from detonation points. This is the most accurate method, but is only valid if the straight line paths are through the explosive material.
>
> **Indirect Path** : Detonation times are calculated from shortest paths through explosive material. Paths are automatically computed around inert materials (e.g. wave shapers).

Plot detonation points

Check this box if you want to see the location of detonation points in the view panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (2D) - Point



This window lets you define a detonation point.

X/Y Coordinates.

Enter the X/Y/Z coordinates of the detonation point in these fields.

Compute Initiation Time?

Select whether you want to specify a detonation time, or have it computed by AUTODYN.

If selected, AUTODYN will compute the detonation time using paths through the explosive from other detonation points.

You must specify the initiation time for at least one detonation point, plane, cylinder or sphere.

Initiation Time

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the initiation time in field (unless you chose to have it computed).

Limit range of application?

Check this box if you want to limit the range of application of the detonation point (extra dialog is presented).

Range will Include / Exclude region.

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

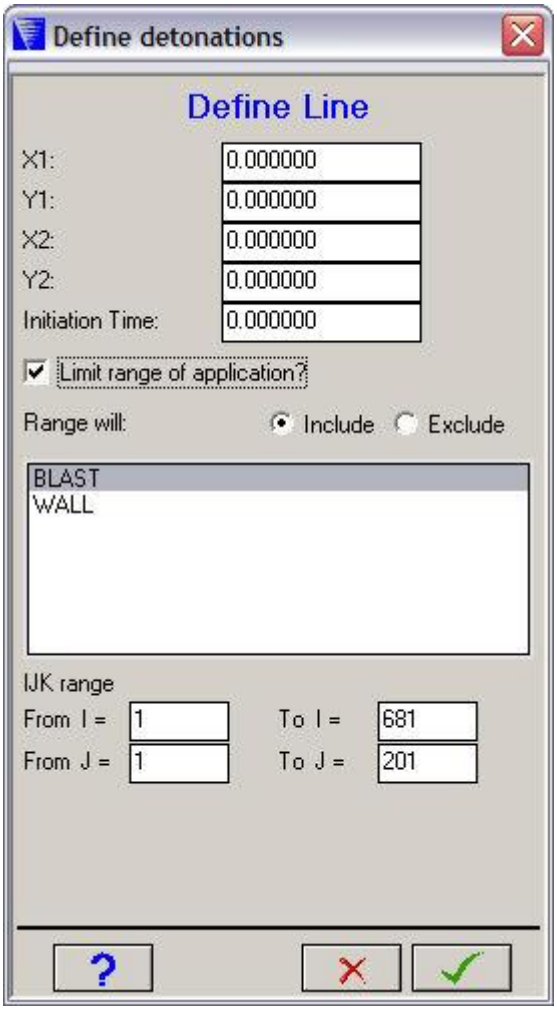
This window lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJ range

Enter the IJ range you want to use to limit the range of application of the detonation point in the fields provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (2D) - Line



This window lets you define a line of detonation.

X/Y Coordinate.

Enter the X and Y coordinates for the two end points of the line.

Initiation Time

Enter the initiation time in field (unless you chose to have it computed).

Limit range of application?

Check this box if you want to limit the range of application of the detonation point (extra dialog is presented).

Range will Include / Exclude region.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

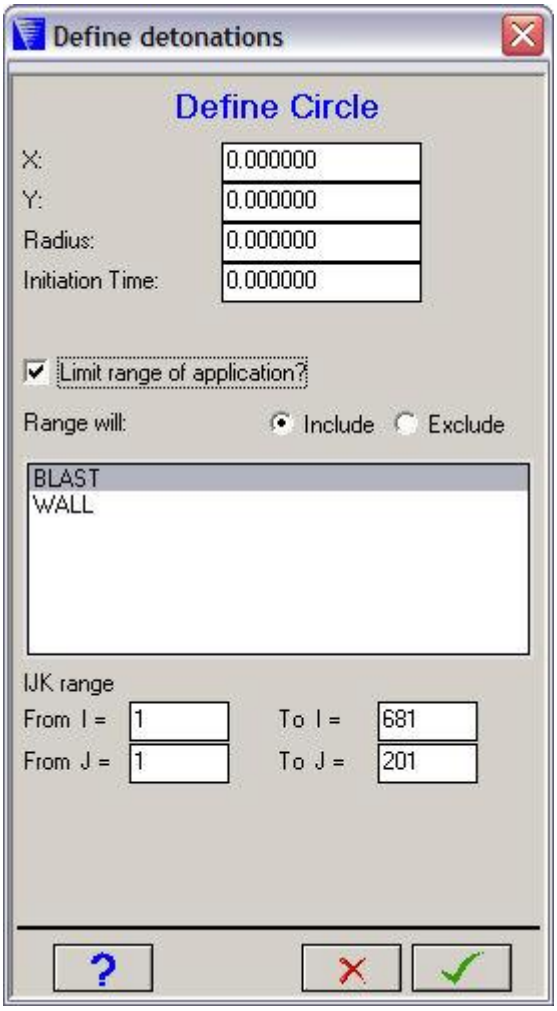
This window lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJ range

Enter the IJ range you want to use to limit the range of application of the detonation point in the fields provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (2D) - Circle



This window lets you define a detonation circle (detonation propagates radially inward and outward from the surface of the circle).

Center (X,Y)

Enter the X and Y coordinates of the center of your circle in the fields provided.

Radius

Enter the radius of your circle in this field.

Initiation Time

Enter the initiation time in field (unless you chose to have it computed).

Limit range of application?

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box if you want to limit the range of application of the detonation point (extra dialog is presented).

Range will Include / Exclude region.

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

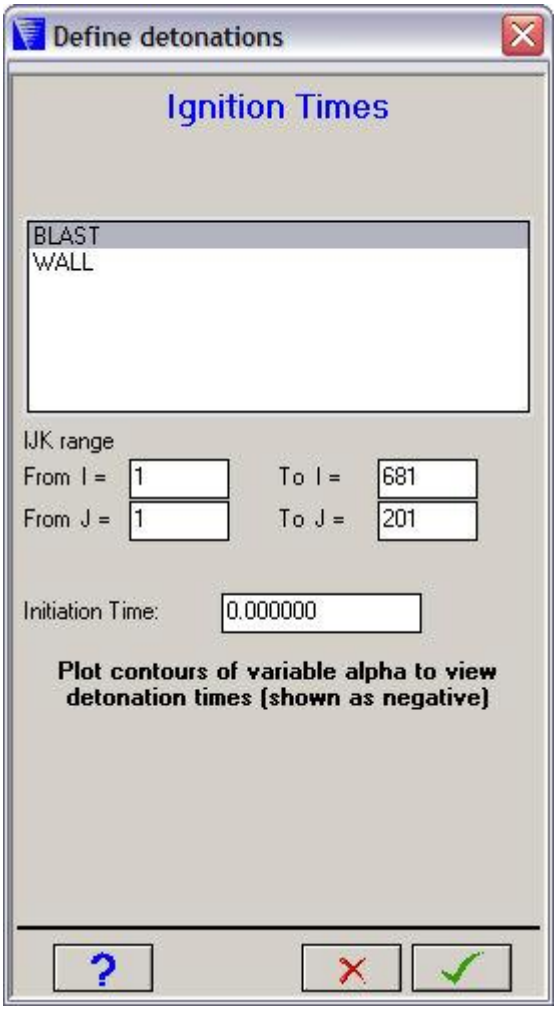
This window lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJ range

Enter the IJ range you want to use to limit the range of application of the detonation point in the fields provided

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (2D) - Manual



This window lets you manually define initiation times for blocks of elements.

Part List

The top box lists all the Parts in your model. Select the Part you want to define initiation time for in this list.

IJ range

Specify the IJ range for a the block of elements to which you want to apply an initiation time.

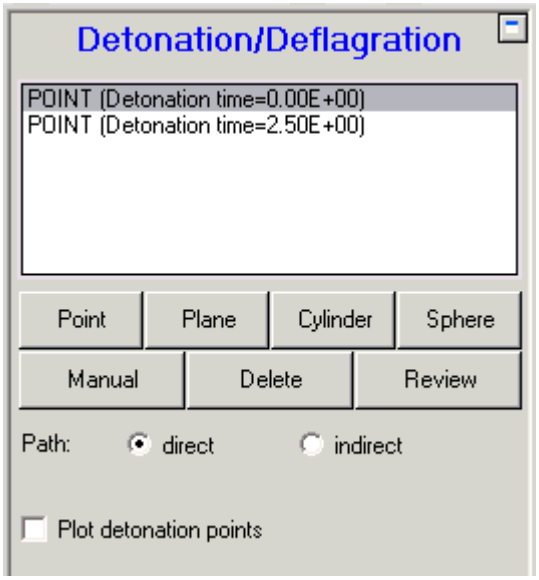
Initiation time

Enter the initiation time that will be applied to all elements in the block in this field.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (3D)

Detonation (3D)



This window lets you set the locations where explosives will be detonated / deflagrated.

Detonations / Deflagrations

The box at the top of the window shows you a list of the current detonations / deflagrations you have defined. You can select from this list.

Point

Press this button to define a detonation point.

Plane

Press this button to define a detonation plane.

Cylinder

Press this button to define a detonation cylinder.

Sphere

Press this button to define a detonation sphere.

Manual

Press this button to manually define initiation times for blocks of elements..

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Delete

Press this button to delete the selected detonation definition

Review

Press this button to review information for all detonation definitions.

Path

Select whether you want detonation times to be calculated using direct or indirect paths.

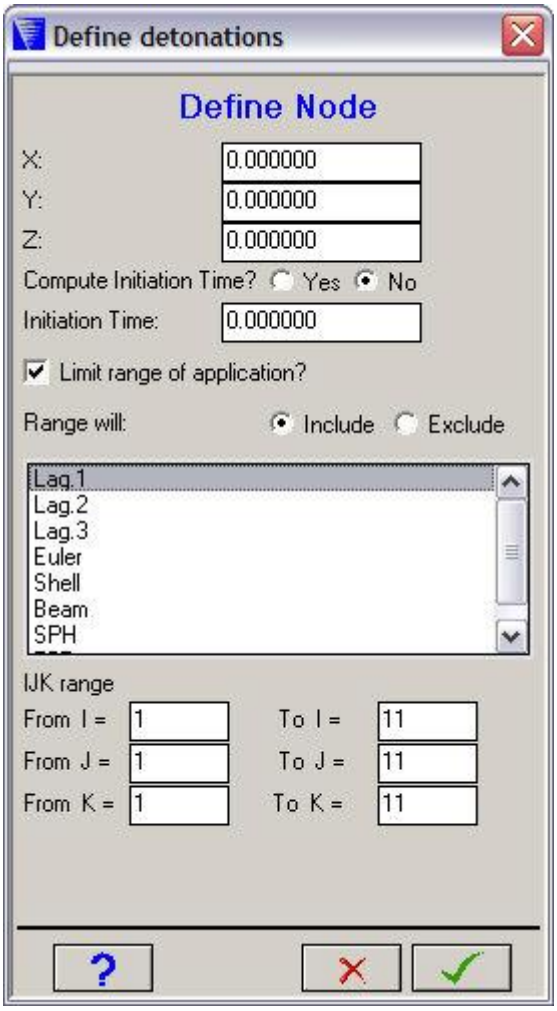
> **Direct Path** : Detonation times are computed from straight line paths from detonation points. This is the most accurate method, but is only valid if the straight line paths are through the explosive material.
>
> **Indirect Path** : Detonation times are calculated from shortest paths through explosive material. Paths are automatically computed around inert materials (e.g. wave shapers).

Plot detonation points

Check this box if you want to see the location of detonation points in the view panel.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (3D) - Point



This window lets you define a detonation point.

X/Y/Z Coordinates.

Enter the X/Y/Z coordinates of the detonation point in these fields.

Compute Initiation Time?

Select whether you want to specify a detonation time, or have it computed by AUTODYN.

If selected, AUTODYN will compute the detonation time using paths through the explosive from other detonation points.

You must specify the initiation time for at least one detonation point, plane, cylinder or sphere.

Initiation Time

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the initiation time in field (unless you chose to have it computed).

Limit range of application?

Check this box if you want to limit the range of application of the detonation point (extra dialog is presented).

Range will Include / Exclude region.

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

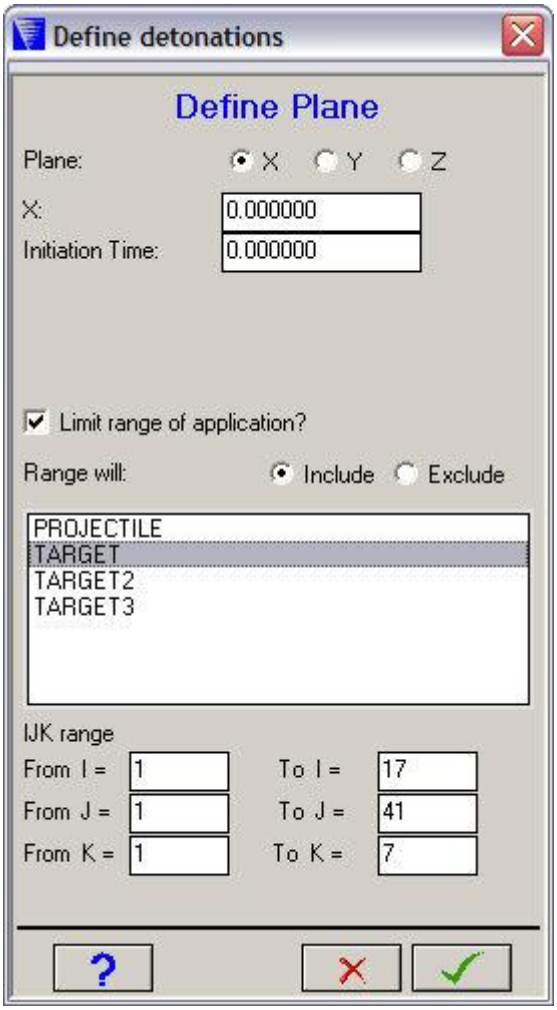
This window lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJK range

Enter the IJK range you want to use to limit the range of application of the detonation point in the fields provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (3D) - Plane



This window lets you define a detonation plane.

Plane

Select whether you want to define an X, Y or Z detonation plane.

X/Y/Z Coordinate.

Enter the X, Y or Z coordinate of the plane in this field.

Initiation Time

Enter the initiation time in field (unless you chose to have it computed).

Limit range of application?

Check this box if you want to limit the range of application of the detonation point (extra dialog is presented).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Range will Include / Exclude region.

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

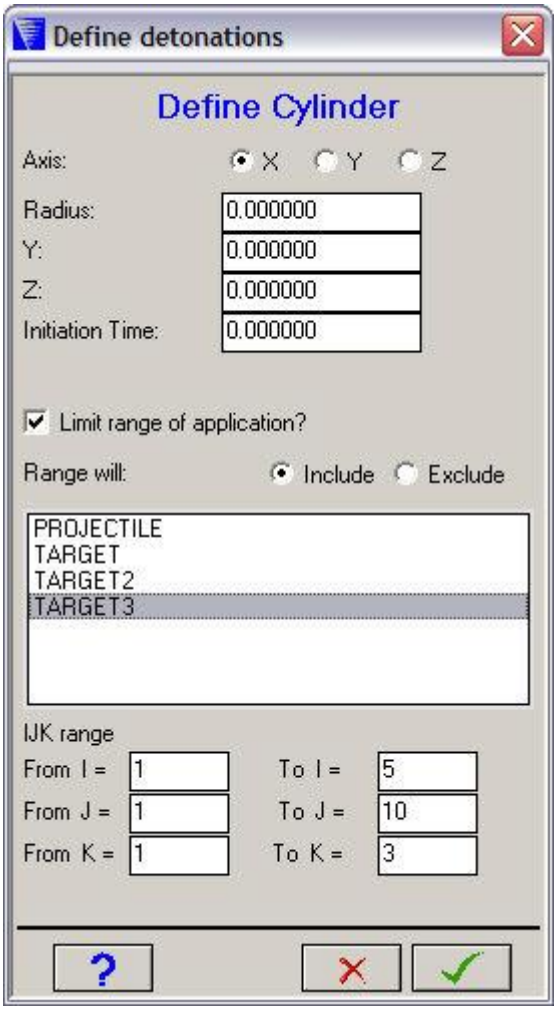
This window lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJK range

Enter the IJK range you want to use to limit the range of application of the detonation point in the fields provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (3D) - Cylinder



This window lets you define a detonation cylinder (detonation propagates radially inward and outward from the surface of the cylinder).

Axis

Select the axis (X, Y or Z) to which the axis of symmetry of your cylinder will be parallel.

Radius

Enter the radius of your cylinder in this field.

Center Coordinates.

Enter the two coordinates for the center of your cylinder in the fields provided.

Initiation Time

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the initiation time in field (unless you chose to have it computed).

Limit range of application?

Check this box if you want to limit the range of application of the detonation point (extra dialog is presented).

Range will Include / Exclude region.

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

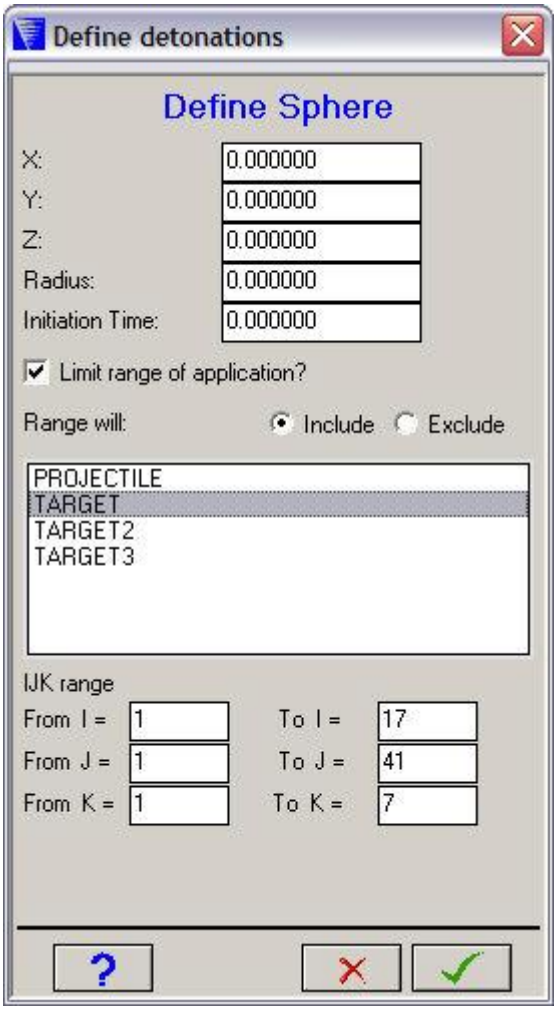
This window lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJK range

Enter the IJK range you want to use to limit the range of application of the detonation point in the fields provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (3D) - Sphere



This window lets you define a detonation sphere (detonation propagates radially inward and outward from the surface of the sphere).

Center Coordinates.

Enter the coordinates for the center of your sphere in the fields provided.

Radius

Enter the radius of your sphere in this field.

Initiation Time

Enter the initiation time in field (unless you chose to have it computed).

Limit range of application?

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Check this box if you want to limit the range of application of the detonation point (extra dialog is presented).

Range will Include / Exclude region.

For each Part, you can specify an index range that is used to limit the range of application of the detonation point.

Select whether you want the index range to specify the included region or the excluded region.

Part list

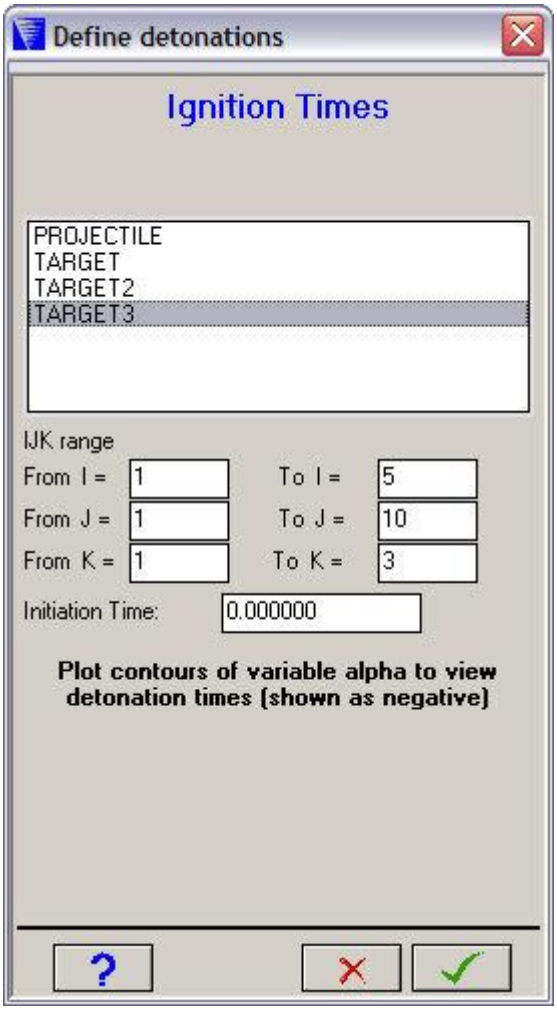
This window lists all the Parts in your model. Select a Part in this window to limit its range of application.

IJK range

Enter the IJK range you want to use to limit the range of application of the detonation point in the fields provided.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Detonation (3D) - Manual



This window lets you manually define initiation times for blocks of elements.

Part List

The top box lists all the Parts in your model. Select the Part you want to define initiation time for in this list.

IJK range

Specify the IJK range for a block of elements to which you want to apply an initiation time.

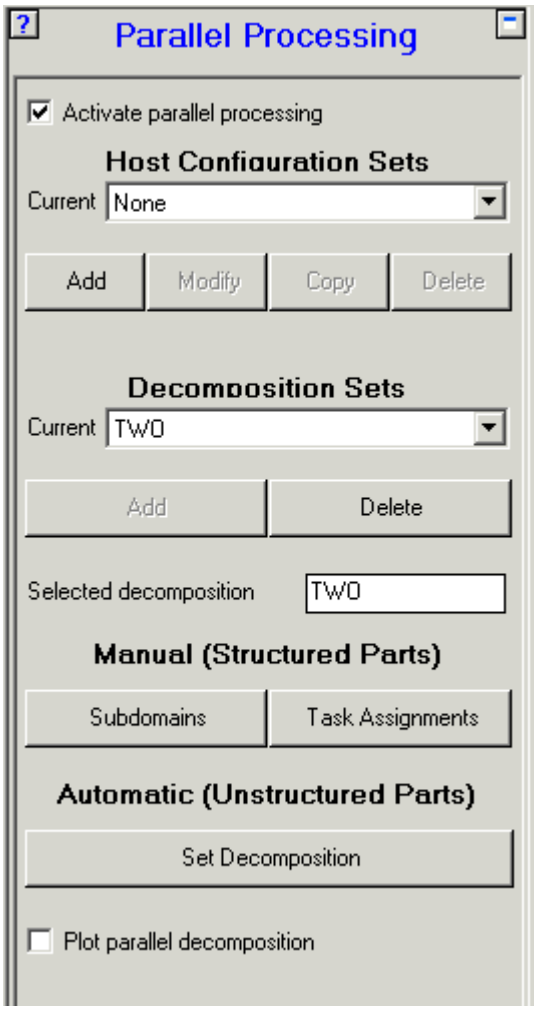
Initiation time

Enter the initiation time that will be applied to all elements in the block in this field.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parallel

Parallel



This panel lets you set up your model for parallel processing For more additional information about setting up a parallel analysis see also the AUTODYN Parallel Processing Tutorial Version 12.0.

Manual (Structured Parts)

The structured Lagrangian solver is decomposed along its grid lines. These provide an easy and obvious method to decompose any structured Parts whilst maximizing the load balancing and minimizing the communication.

Activate parallel processing
Check this box to activate parallel processing. If this box is checked, your model will be executed in parallel when you press

Host Configuration Sets

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This input lets you configure the hardware you want to use for parallel processing by defining one or more Host Configuration Sets.

These configuration sets are saved independently of your model. Any sets you create will automatically be available for other models you have set up.

The box at the top shows the name of the current Host Configuration Set. The pull-down menu for this box lets you change the current Host Configuration Set to any other Set you have defined.

Add

Press this button to define a new Host Configuration Set.

Modify

Press this button to modify the current Host Configuration Set.

Copy

Press this button to copy an existing Host Configuration Set to a new Host Configuration Set.

Delete

Press this button to delete the current Host Configuration Set.

Decomposition Sets

This input lets you define the domain decompositions you want to use for parallel processing by defining one or more Decomposition Sets.

Unlike Host Configuration Sets, Decomposition Sets are specific to your model.

The box at the top shows the name of the current Host Decomposition Set. The pull-down menu for this box lets you change the current Decomposition Set to any other Set you have defined.

Add

Press this button to define a new Decomposition Set. Enter the name of your new Decomposition Set in the **Selected decomposition** field.

Delete

Press this button to delete the current Decomposition Set.

Subdomains

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press this button to decompose the Parts of your model into sub-domains (SPH parts are decomposed automatically).

Task Assignments

Press this button to assign the sub-domains to Tasks performed on each CPU.

Automatic (Unstructured Parts)

The automatic domain decomposition for unstructured parts is conducted using a recursive bisection method. This takes into account the geometric location of elements, the element type and the element density.

Set Decomposition

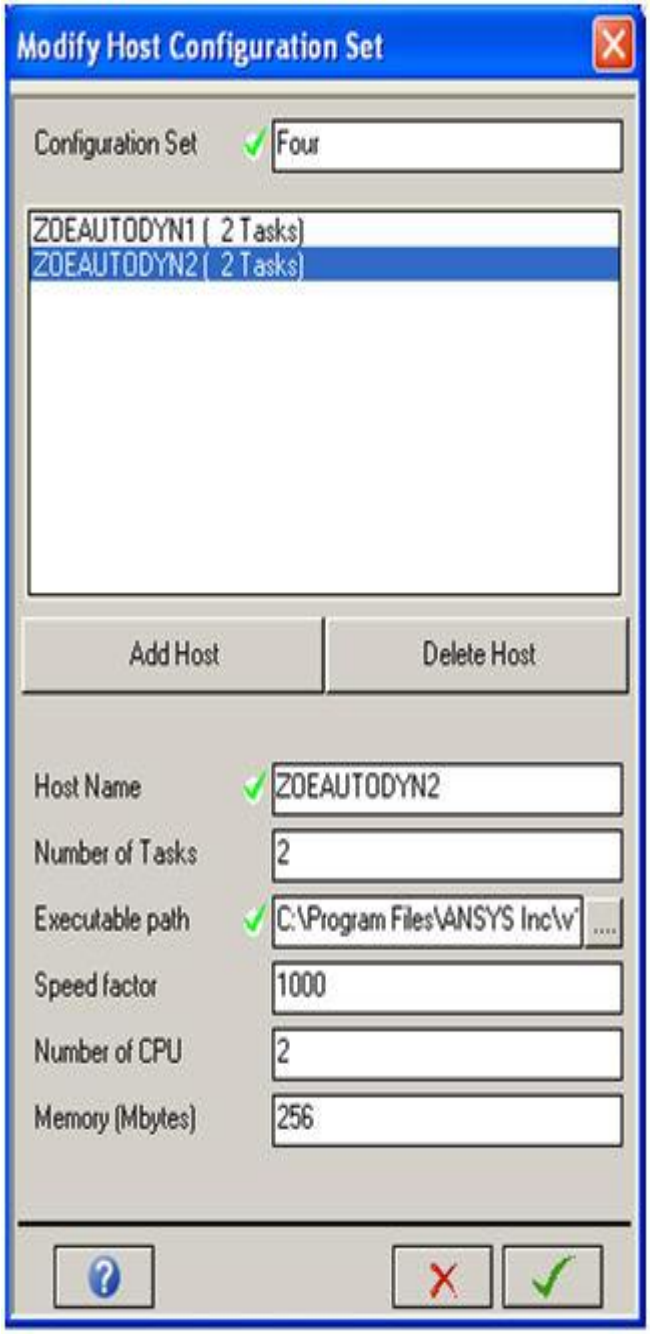
Press this button to select the parts to be included in decomposition.

Plot parallel decomposition

Press this button to see the domain decomposition and task assignments for the current Decomposition Set.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parallel - Add / Modify



This window lets you create a new Host Configuration or modify an existing Host Configuration Set

Configuration Set

If you are creating a new Host Configuration Set, enter its name in this field.

If you are modifying an existing Host Configuration Set, the name of the current configuration set is displayed in this field.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

List of Hosts

The large box lists all the Host machines currently included in this Host Configuration Set. You can select a host in this list.

Add Host

Press this button to add a host to this Host Configuration Set (the Host Data fields are cleared to let you input data for the new host).

Delete

Press this button to delete the selected host from this Host Configuration Set.

Host Data

Host Name

Enter the name of the host machine in this field

Number of Tasks

Enter the number of tasks to be run on this host. If you set the number of tasks equal to the number of CPUs, one task will automatically be run on each CPU (this is usually the most efficient way to run parallel calculations).

Executable path

Enter the path where the AUTODYN executables are located (usually in a shared directory).

Speed factor

If the hosts in your Host Configuration Set have different speeds, set the speed factor for each host in this field.

If all hosts have the same speed, set all speed factors to the same value (any value is OK).

Number of CPU

Enter the number of CPUs on the host in this field.

Memory

Enter the amount of memory on the host in this field (this information is not currently used).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parallel - Copy

This window lets you copy an existing Host Configuration Set to a new or existing Host Configuration Set.

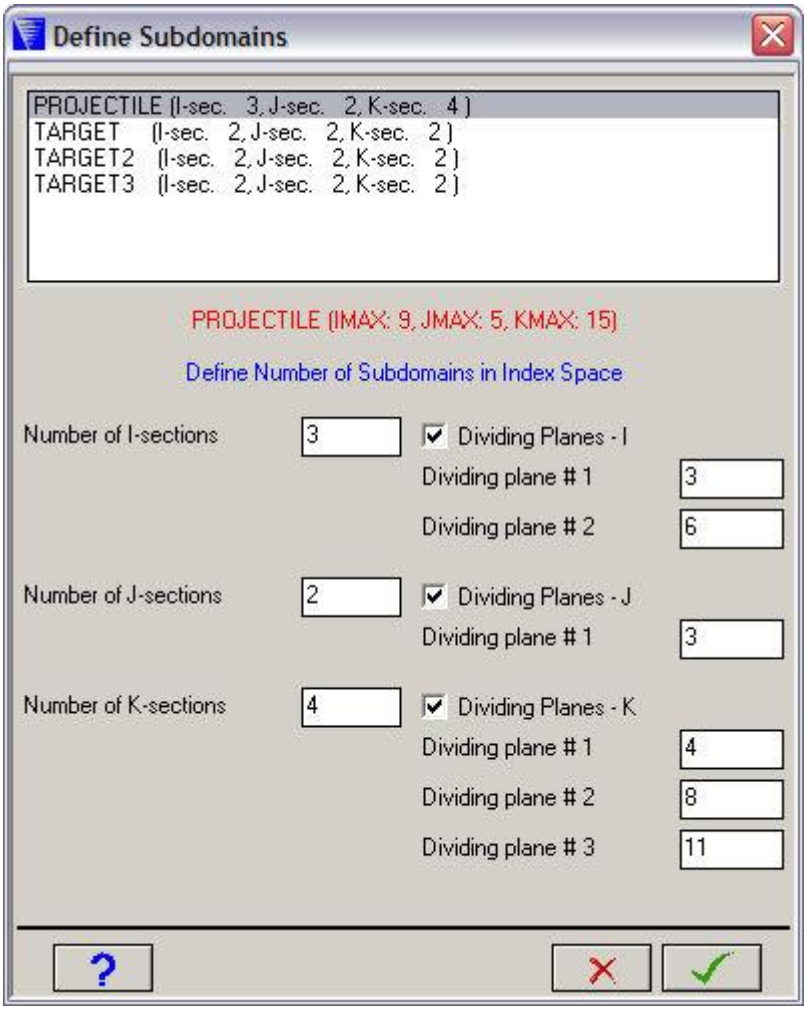
Select the Host Configuration Set you want to copy in the top pull-down menu.

In the lower pull-down menu, select the Host Configuration Set to which you want to copy the data (to copy to a new Configuration Set, select <New>, then, in the Parallel Processing panel, rename the Configuration Set to the name you want to use).

Press to perform the copy.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parallel - Sub-domains



This window lets you decompose the Parts of your model into sub-domains for parallel processing (SPH parts are decomposed automatically).

You can decompose each grid-based Part in you model into smaller blocks, called sub-domains, that can be assigned to different tasks (on different hosts) for processing.

For each Part, these blocks are formed by defining planes in the I, J and K direction where cuts are made to form the sub-domains.

Parts List

The top box lists the model Parts that you can decompose for parallel processing. You can select a Part in this list.

Number of Sections

Enter the number of sections into which you want the selected part divided in I, J and K directions.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

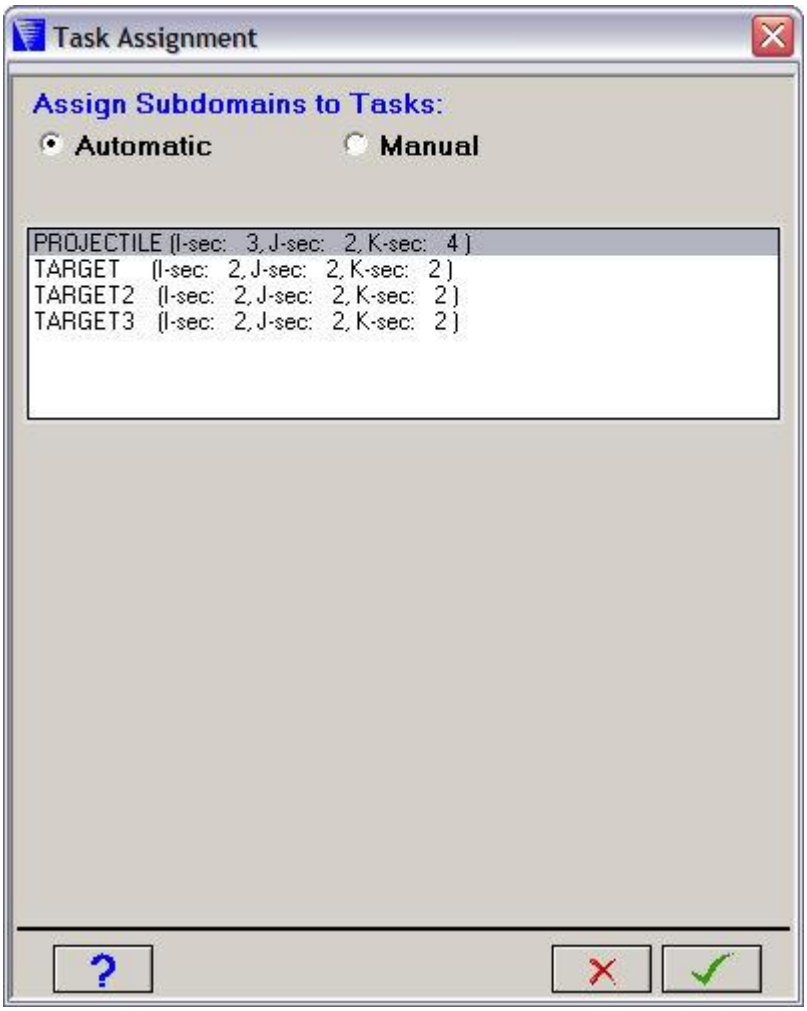
Dividing Planes

Enter the I, J and K values for the dividing planes in the three index directions (the default values place the dividing planes evenly through the grid).

The example above would decompose the **PROJECTILE** Part into 24 sub-domains

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parallel - Task Assignment


Task Assignment
Assign Subdomains to Tasks:
Automatic
Manual
PROJECTILE (I-sec: 3, J-sec: 2, K-sec: 4)
TARGET (I-sec: 2, J-sec: 2, K-sec: 2)
TARGET2 (I-sec: 2, J-sec: 2, K-sec: 2)
TARGET3 (I-sec: 2, J-sec: 2, K-sec: 2)
?

This window lets you assign sub-domains to Tasks performed on each CPU.

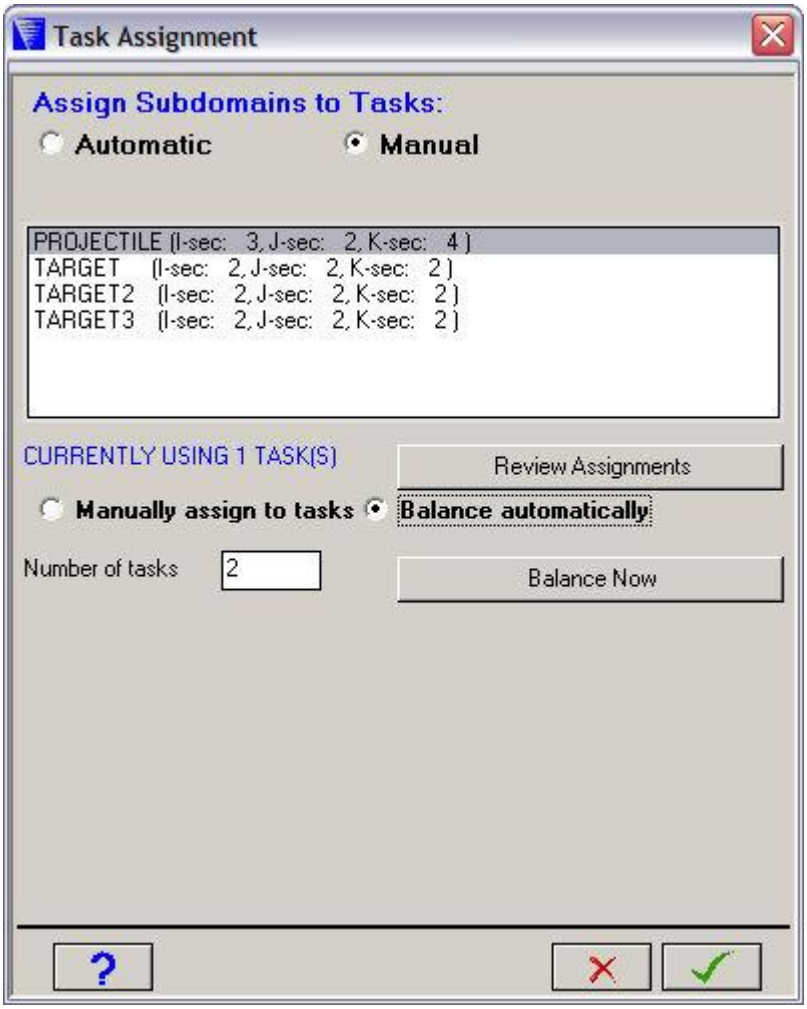
The efficiency of parallel processing is very dependent on distributing sub-domains among the tasks so that each task has approximately the same amount of work (computing) to do.

Automatic / Manual assignment.

If you select **Automatic** assignment of Sub-domains to Tasks, AUTODYN will automatically distribute sub-domains among the tasks, using load-balancing algorithms. You do not need to input any further data if you use this option.

If you select Manual assignment of Sub-domains to Tasks, further input dialog is presented :

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Parts List

The box at the top of the window lists the Parts whose Sub-domains you need to assign. You can select a Part in this list.

Review Assignments

Press this button to view a list of all Sub-domains and the tasks to which they are assigned.

Balance Automatically

If you select this option, you can invoke the load-balancing algorithms use if you had selected **Automatic** assignment of the Sub-domains to Tasks, to distribute your Sub-domains over a specific number of Tasks.

Number of tasks

Enter the number of Tasks over which you want to distribute your Sub-domains.

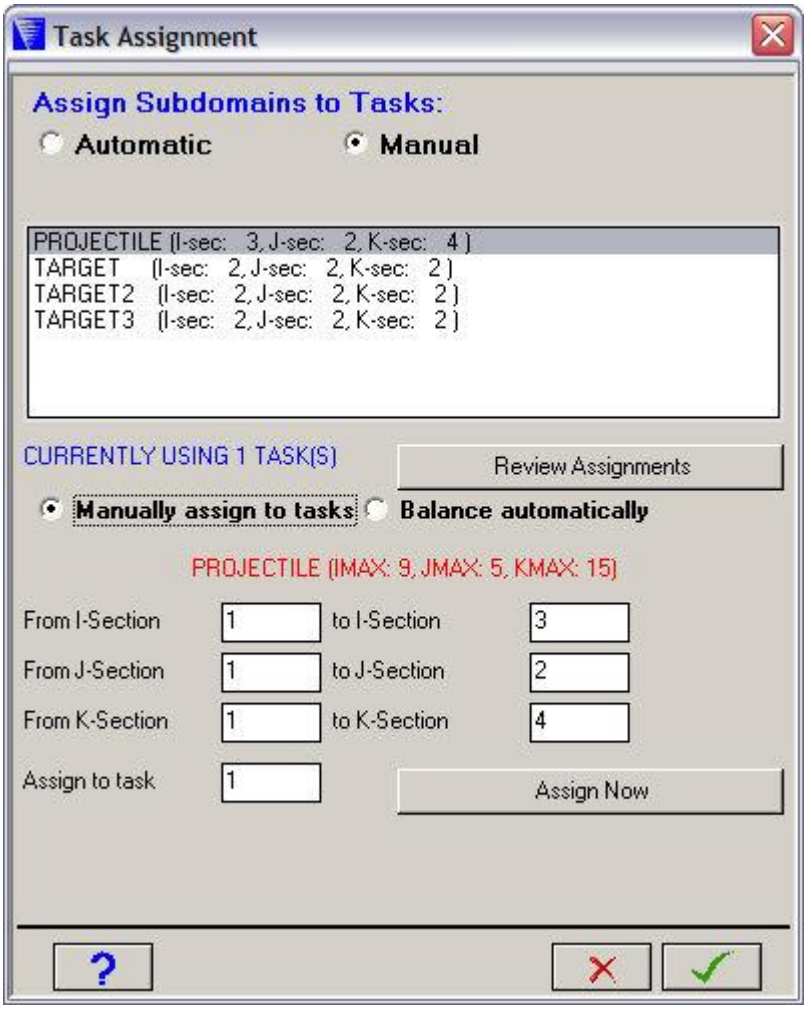
Balance Now

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Press this button to perform the distribution.

Manually assign to tasks

If you select this option, dialog is presented which lets you manually assign blocks of Sub-domains to a specific Task :



Block to be assigned

In the fields provided, specify the I, J and K section range for the block of Sub-domains you want to assign to a Task.

Assign to Task

Enter the Task # to which you want to assign the block of Sub-domains in this field.

Assign Now

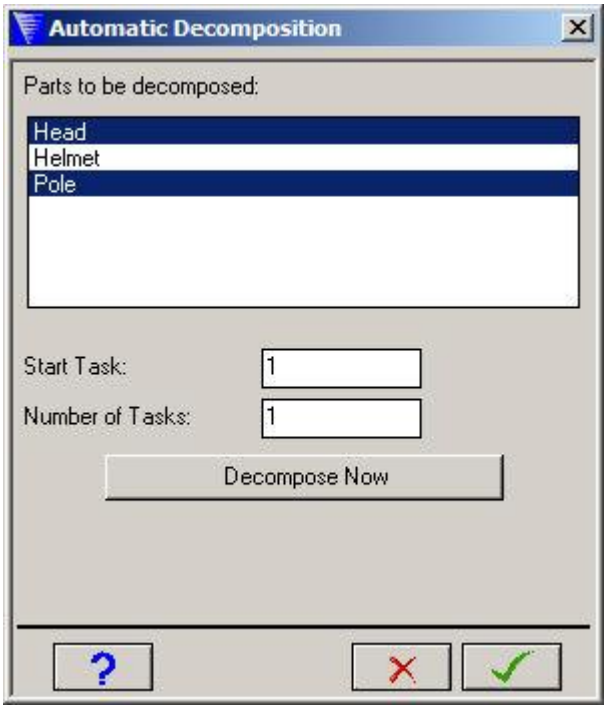
Press this button to assign the specified block to the specified Task.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

This procedure can be repeated as many times as you want, to assign blocks of Sub-domains to Tasks.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parallel Automatic Decomposition



Parts to be decomposed

Select the parts to be decomposed from the list of unstructured parts (by default all unstructured parts will be selected).

Start Task

Enter the start task.

Number of tasks

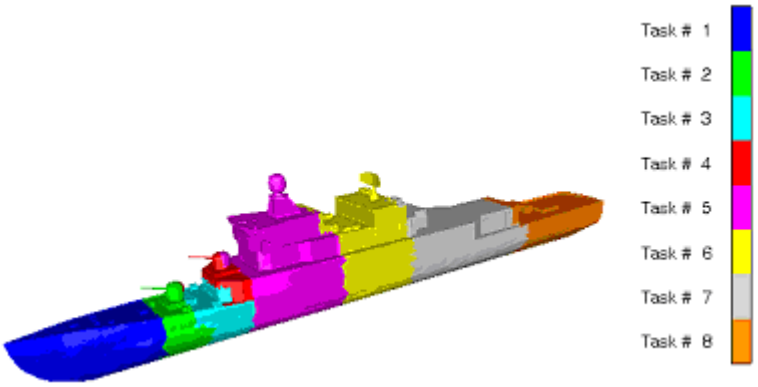
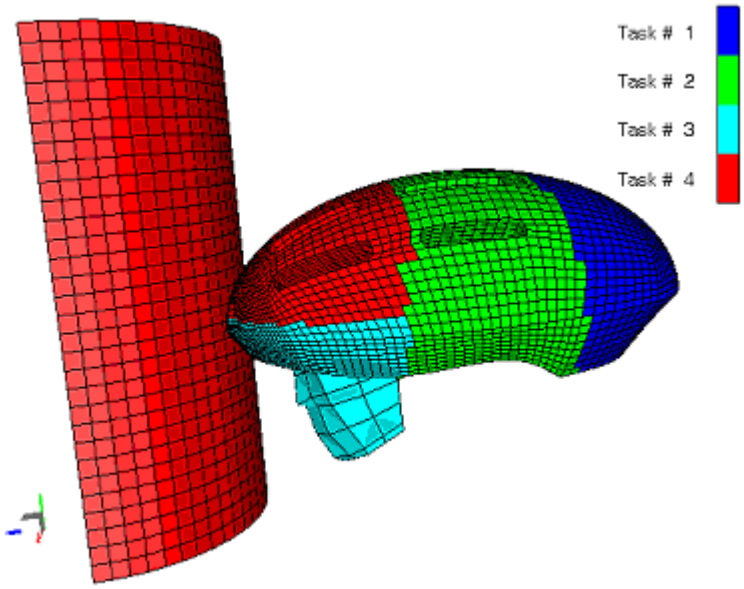
Enter the number of tasks

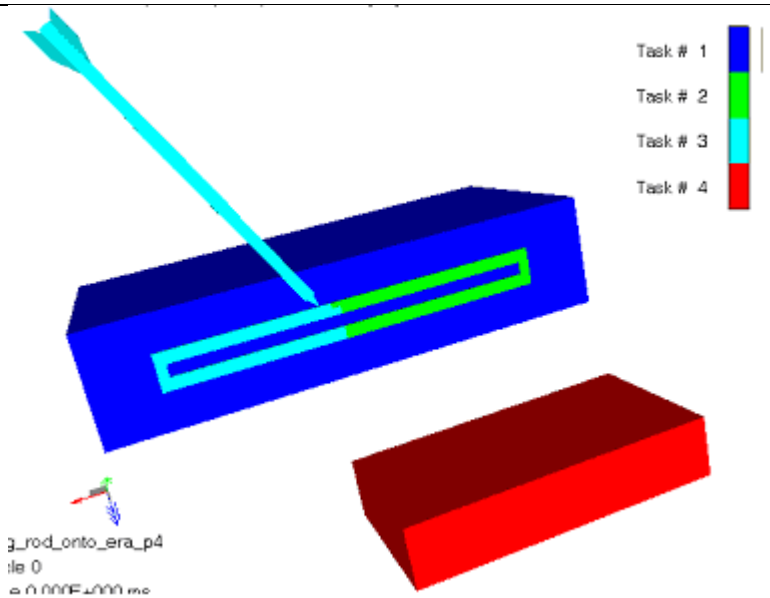
Decompose now

Press the *Decompose Now* button to automatically define the task on which each of the elements contained in the selected Parts will be assigned.

Examples

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Single unstructured Part • Select Part • Start Task = 1 • Number Tasks = 8	
All Parts Unstructured One rigid material • Select All Parts • Start Task = 1 • Number Tasks = 1	

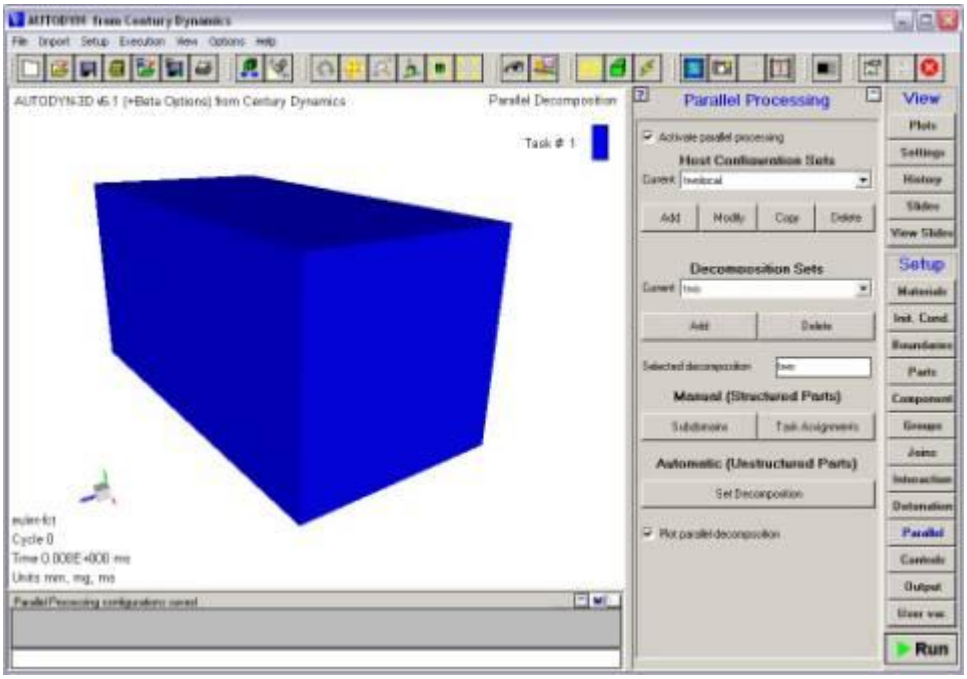
3 Unstructured Parts
 	
1 Euler Part (Structured)
 	
• Manual assignment of Euler Part to Task 1
• Automatic assignment of 2 unstructured Parts (rod and first target)
  ○ Start Task = 2
  ○ # Task = 2
Automatic assignment of unstructured Part (second target)
  ○ Start Task = 4
  ○ # Tasks = 1
	

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Automatic decomposition of Euler-FCT parts

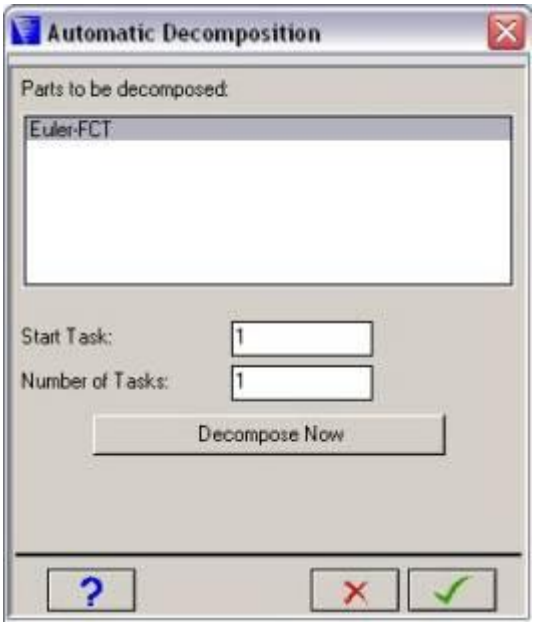
The automatic decomposition algorithm has been extended to incorporate Euler-FCT parts in addition to its present ability to handle unstructured grids.
This facilitates the decomposing of complicated Euler-FCT/Lagrange coupled models – the user need only stipulate the number of tasks over which the model should be assigned and AUTODYN will automatically produce a decomposition configuration with good load balancing qualities and minimal inter-processor communication. Furthermore, to further enhance the efficiency of coupled calculations the subdomains of the Lagrange structure will be placed on the same processor, so decreasing the necessary inter-processor communication.

To automatically decompose an Euler-FCT model, go to the Parallel dialog panel as seen below, activate parallel processing and add a new decomposition set which in this instance is called two:

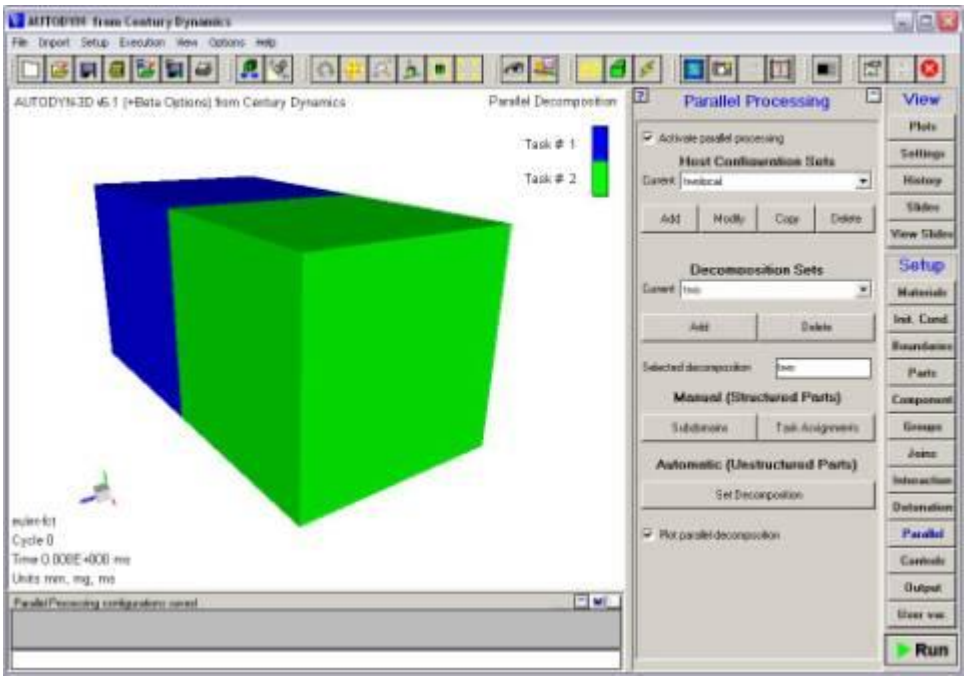


Press the **Set Decomposition** button and all Euler-FCT parts will be shown in the **Automatic Decomposition** dialog window:

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

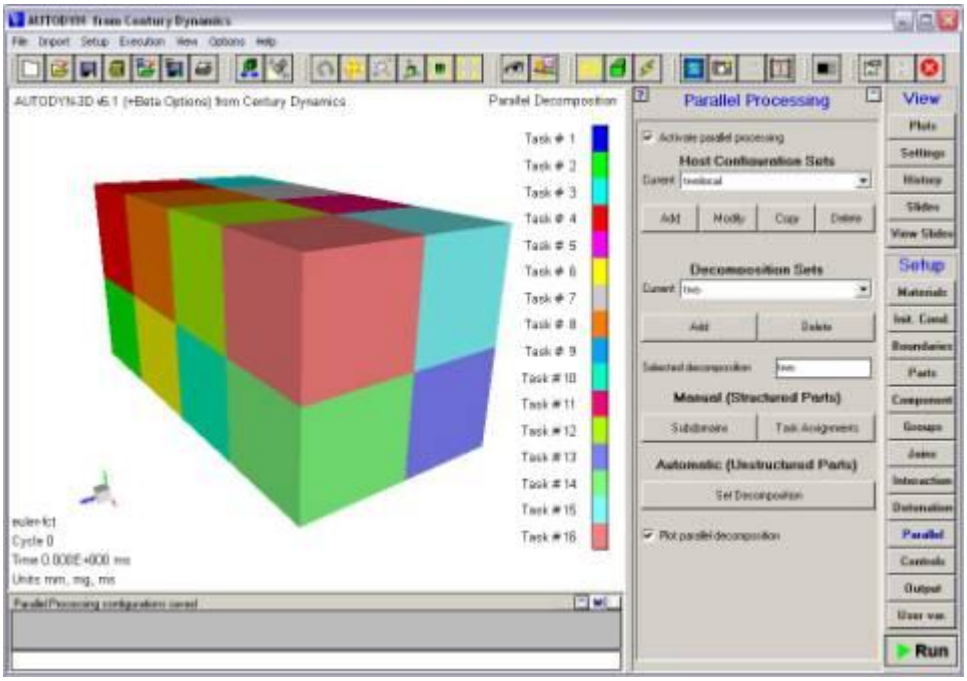


Input the number of tasks you want to decompose the part over and press the **Decompose Now** button. In this case the number of tasks is two and the following decomposition can be viewed:

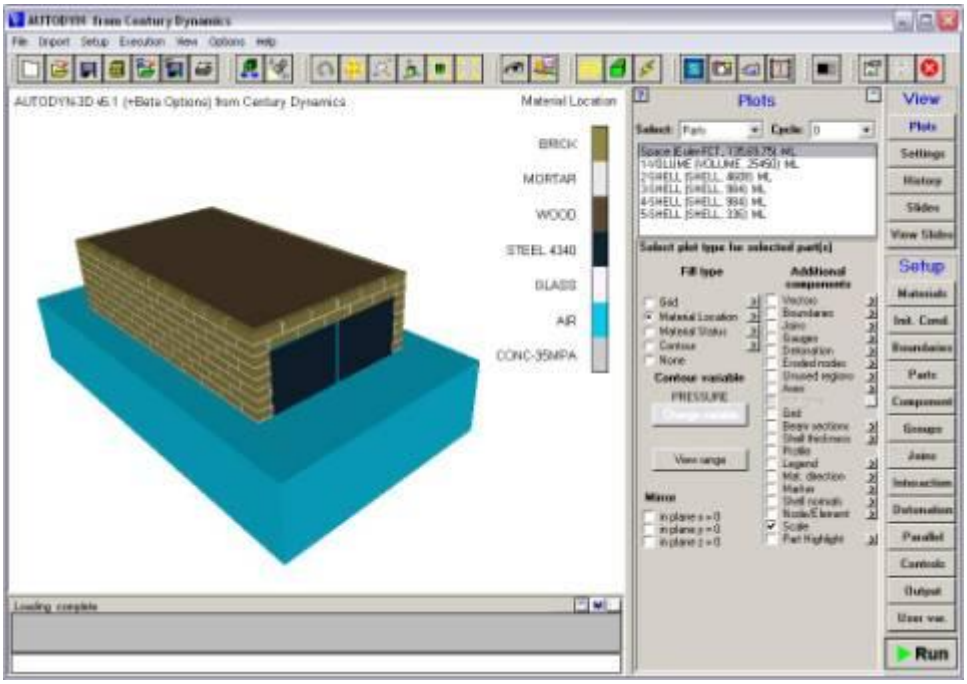


More complicated decompositions can also be easily achieved. Here the part is decomposed over 16 tasks:

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

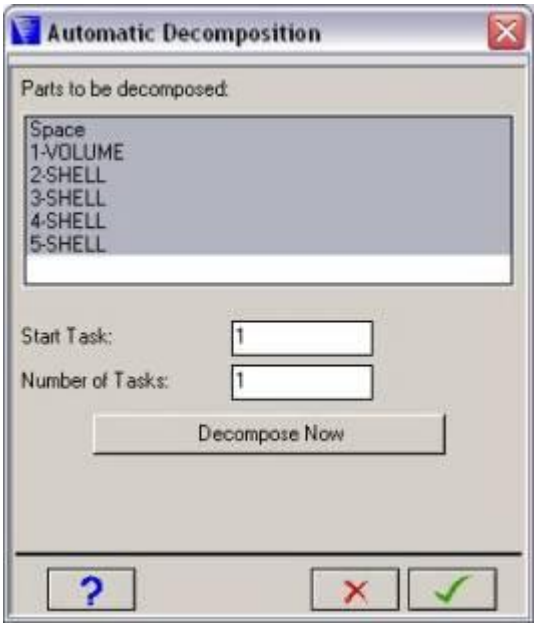


As previously stated the new algorithm allows easy decomposition of complicated coupled models which include Euler-FCT and unstructured Lagrange Parts. The model below contains a brick structure in which an explosion is occurring. The brick structure is modeled using Lagrangian parts, whilst the explosion in air is modeled using an Euler-FCT part. Here, only part of the Euler-FCT part is shown for illustration purposes, the Euler-FCT part extends above the brick structure.

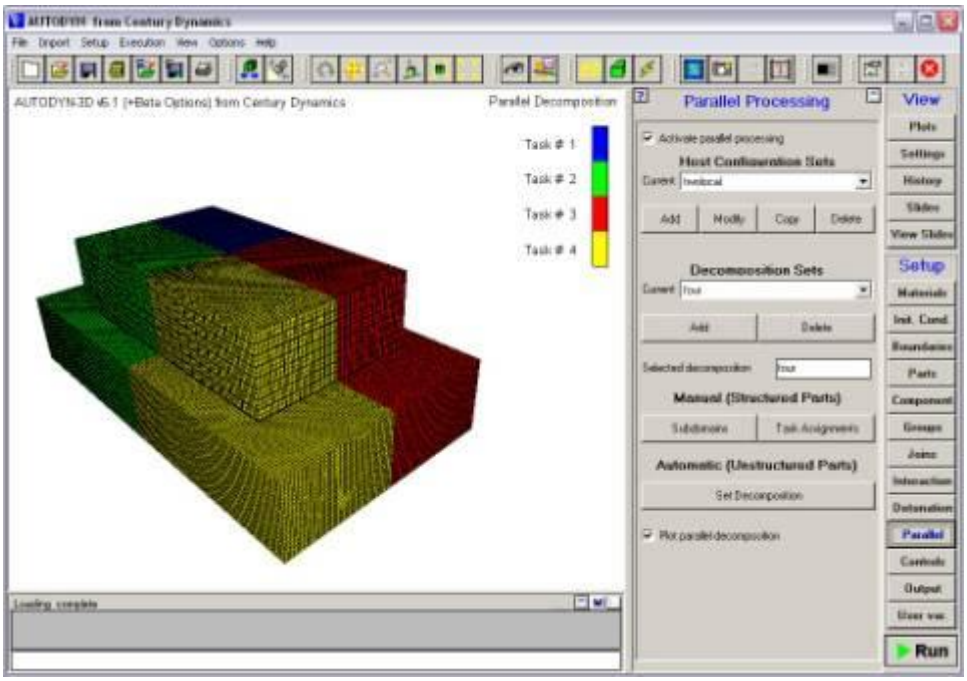


Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The **Decomposition Set** window dialog will contain all of the parts and by default they will all be highlighted:



By choosing decomposing the model over 4 tasks, the various parts can be assigned to the available processors whilst ensuring efficient parallel computations:



If the user wishes to override this decomposition then the Euler-FCT part can still be manually assigned as before.

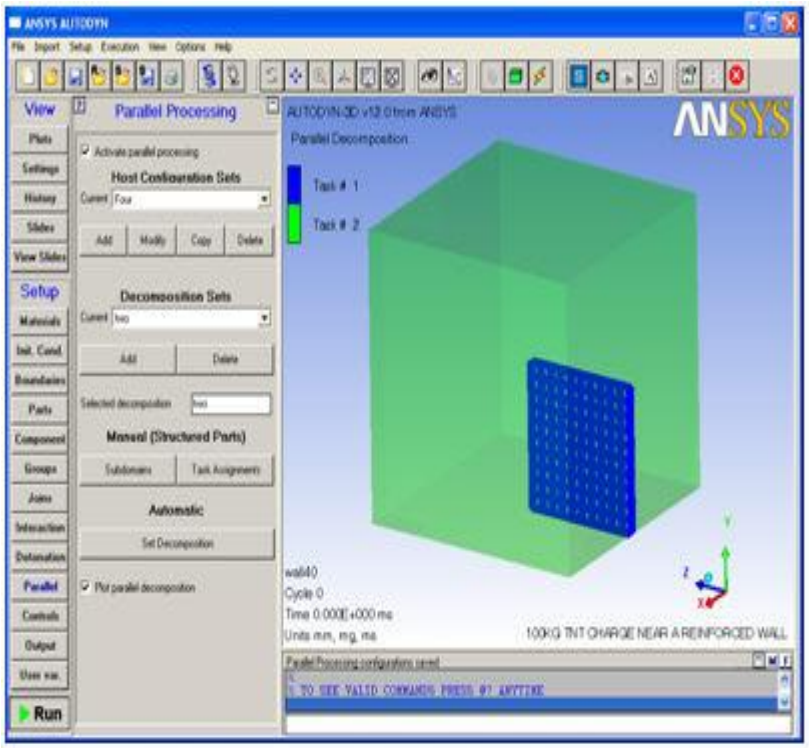
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Parallel - Decomposition of beam parts

AUTODYN-3D simulations including beam elements can be run in parallel.

Structured Beams

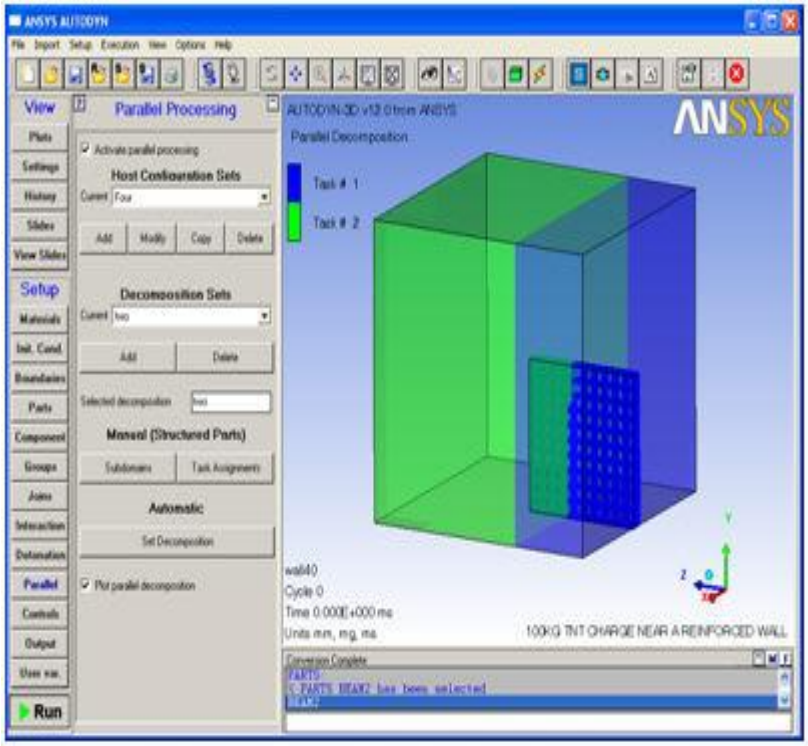
For the parallel computation, each structured Beam Part is considered as a single sub-domain and can be assigned to a specific task for the parallel simulation. Note that individual structured Beam Parts cannot be decomposed into multiple sub-domains.



Unstructured Beams

For parallel computation unstructured Beam Parts can be automatically decomposed, similar to all other unstructured Lagrange Parts.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



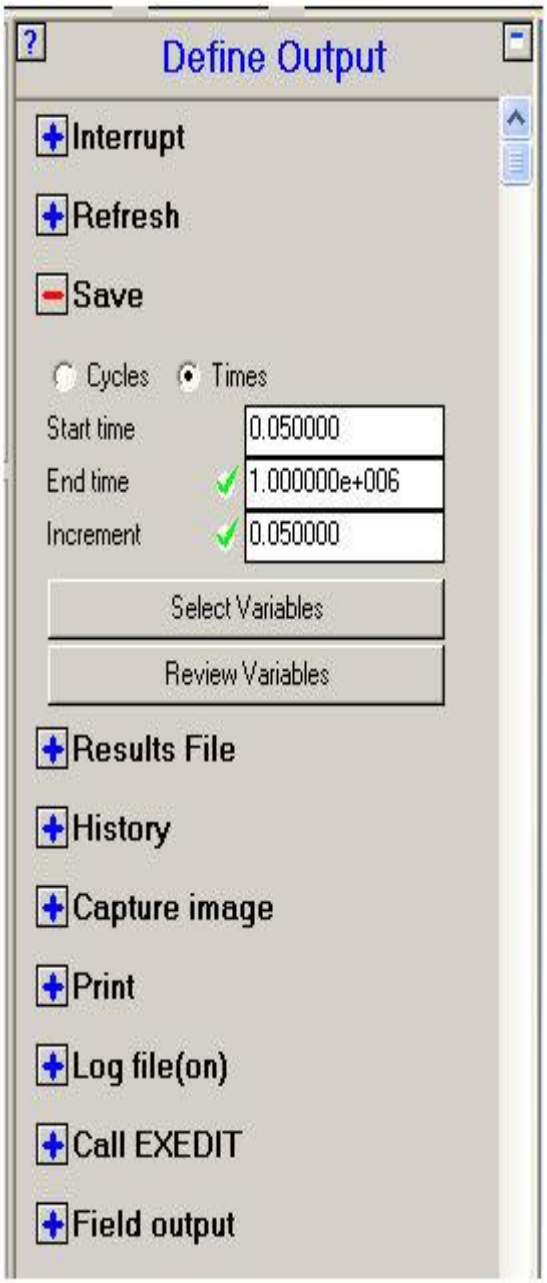
Note that parallel computations including beam elements can currently run with all join and contact options, with the exception of edge on edge contact. This later option is likely to be available in the near future.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Output

Output

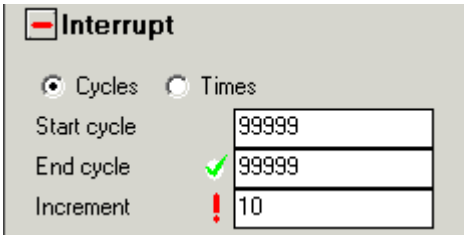
Output



This window lets you set the type of output you want from your calculation.

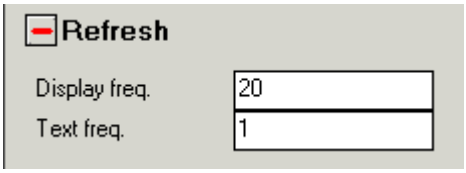
Interrupt

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



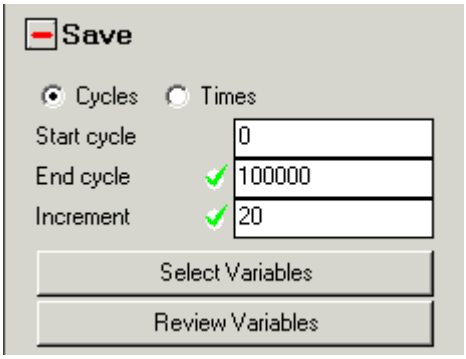
Set the frequency at which your calculation is interrupted (stopped) for you to plot, review and examine.

Refresh



Set the frequency at which your plot in the view panel is refreshed.

Save



The most important type of output you will want is data written to **Save** files.

Cycle / Times

Select whether you want data written at a set cycle frequency or time frequency.

Start cycle / time

Enter the cycle time that you want your first Save file to be written.

End cycle / time

Enter the cycle or time that you want your last Save file to be written.

If you are not sure how long you want to run your problem, enter a large number in this field, then you won't have to come back and change it later.

Increment

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the frequency you wish files to be written between the start cycle / time and the end cycle / time.

Select Variables

Press this button to select the structured or unstructured variables you want written to the **Save** file.

For the structured solvers, the majority of variables is automatically selected and cannot be de-selected. This is because the memory for these variables is always allocated, even if the variable is not being used in a Part.

For the unstructured solvers, the full list of potential variables that can be used is presented. Only a small number of variables are pre-selected as commonly used variables. The selection of additional variables is up to the user and must be made

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

prior to execution of the model (at cycle zero). Only those variables selected will be available for output (contour, examine etc) Note that on execution, if a particular combination of element type and material requires a variable to be saved that has not been selected, AUTODYN will automatically add this to the list.

Also note that the unstructured solves, some of the variables are never allocated physical memory, instead they are reconstructed from other variables that are allocated and stored in memory. An example of this would be the absolute velocity; this can be reconstructed from the nodal velocities hence does not require independent storage.

A similar list is also presented under the History, Variables option.

Note that the selection of additional post-processing variables will result in higher memory usage and file sizes. If a significant number are selected, some degradation in runtime performance can also be expected.

Press + alongside any of the remaining output options to reveal the input fields for that option:

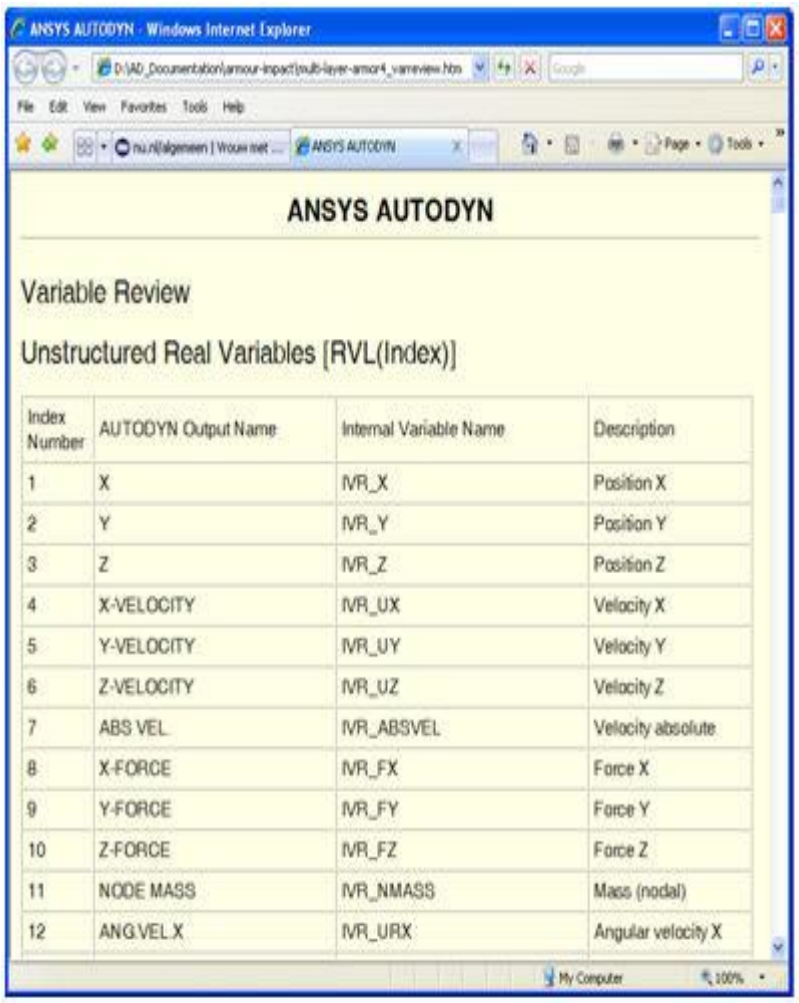
Review Variables

Select this button to create an html page of structured or unstructured variables.

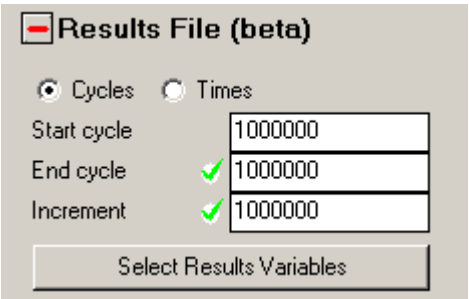
After selection of the Review button first a selection window appears where the type of variable for review needs to be selected, e.g. 2D or 3D, structured or unstructured.

An html page will be created that shows a list of requested variables and their description.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5


ANSYS AUTODYN - Windows Internet Explorer
File Edit View Favorites Tools Help
ANSYS AUTODYN
Variable Review
Unstructured Real Variables [RVL(Index)]
Index Number | AUTODYN Output Name | Internal Variable Name | Description
1 | X | IVR_X | Position X
2 | Y | IVR_Y | Position Y
3 | Z | IVR_Z | Position Z
4 | X-VELOCITY | IVR_UX | Velocity X
5 | Y-VELOCITY | IVR_UY | Velocity Y
6 | Z-VELOCITY | IVR_UZ | Velocity Z
7 | ABS VEL | IVR_ABSVEL | Velocity absolute
8 | X-FORCE | IVR_FX | Force X
9 | Y-FORCE | IVR_FY | Force Y
10 | Z-FORCE | IVR_FZ | Force Z
11 | NODE MASS | IVR_NMASS | Mass (nodal)
12 | ANG.VEL.X | IVR_URX | Angular velocity X

Results File



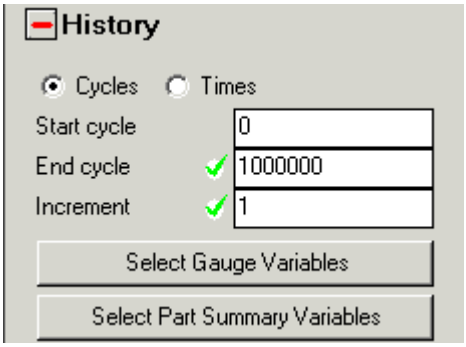
Set the frequency at which results files are written out.

Select Results Variable

Press this button to select the grid variables written to the result file.

History

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Set the frequency at which history data is written.

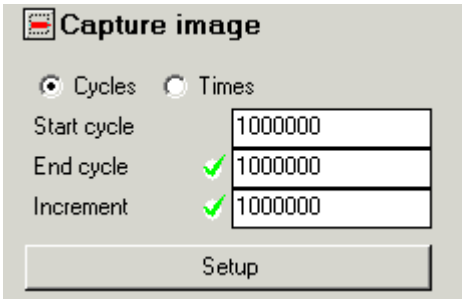
Select Gauge Variables

Select this button to choose which structured and unstructured variables to write to the history data file.

Select Part Summary Variables

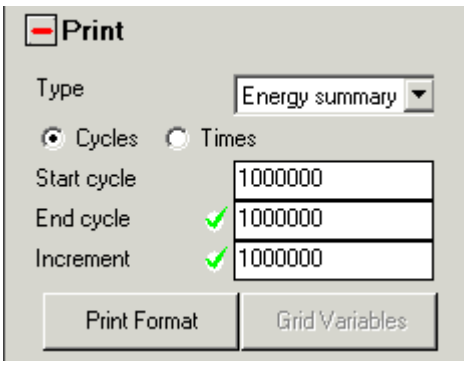
Select this button to choose which part summary variables to write to the history data file.

Capture image



Set the frequency at which automatic image captures are done.

Print



Set the frequency at which printed data is written.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Print Format

Press this button to select the formatting of the information written to the print file.

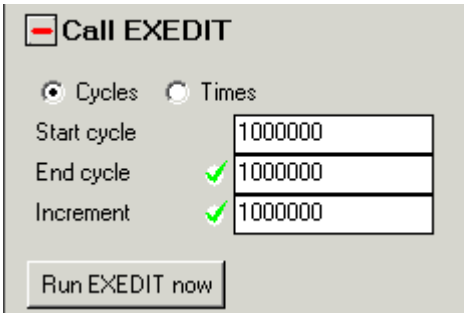
Grid Variables

Press this button to select the grid variables written to the print file.

Log file

Turn the Log file on or off.

Call EXEDIT

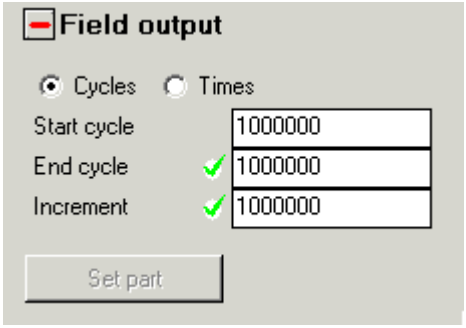


Set the frequency at which subroutine EXEDIT is called.

Run EXEDIT now

Press this button to call the EXEDIT user subroutine

Field output



Set the frequency at which field output is written for weak coupling analyses.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

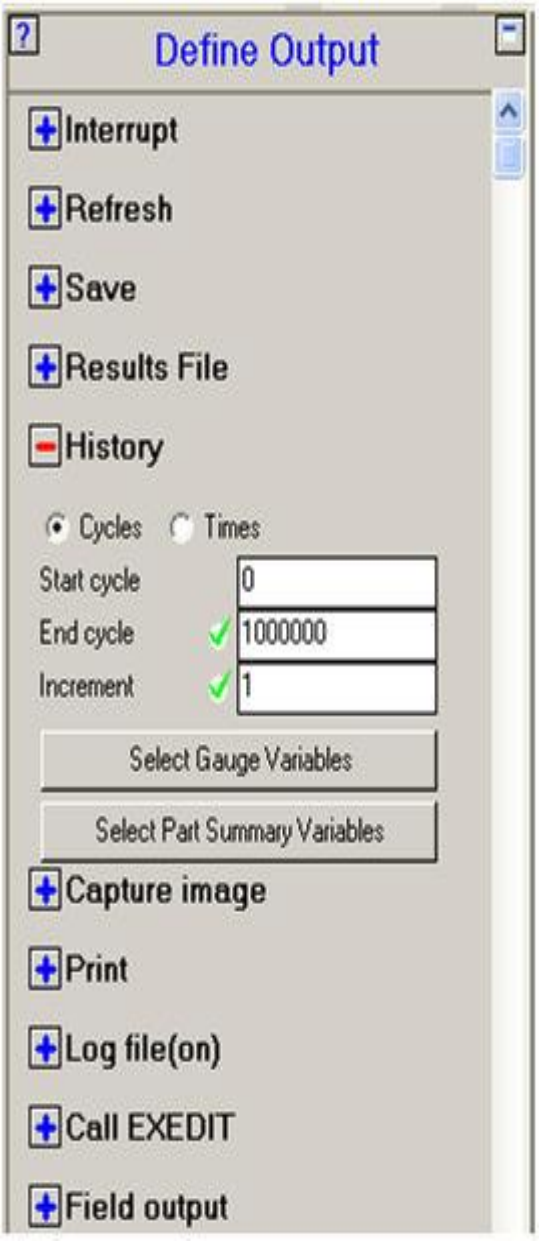
Output-Part Summary Variables

The Part summary variable includes three sets of optional data.

- Contact energy
- Hourglass energy
- External forces

This data is accumulated per Part and is available for review via the print file (using the energy summary option) or as time history data (using the Part summary option).

Note that by default, these additional variables are not available for history plotting. The variables must be activated using the Output, History, Select Part Summary Variables option.



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Hourglass energy

The 2D and 3D Lagrangian volume and shell elements in AUTODYN use hourglass control to remove zero energy modes of deformation. The forces of the hourglass control algorithms dissipate energy in resisting the zero energy modes of deformation.

In AUTODYN, this energy is calculated and accumulated in the element internal energy. In version 6.1, the hourglass energy is now also accumulated independently to the internal energy to allow users to interrogate and assess the relevance of hourglass energy in a simulation.

Contact energy

Contact energy is accumulated on a per part basis as

$$E^{n+1}_{npart} = E^{n}_{npart} + \sum_{1,Nodes\,on\,npart} F^{i}_{contact} \times \Delta x^{i}_{contact}$$

In the absence of friction, the contact energies of two Parts interacting should be equal, but opposite sign, hence have very little contribution to the energy of the overall system. Large imbalances in contact energy are indicative of poor model setups.

For the purpose of energy accountancy, the contact energy is added to the work done on the system.

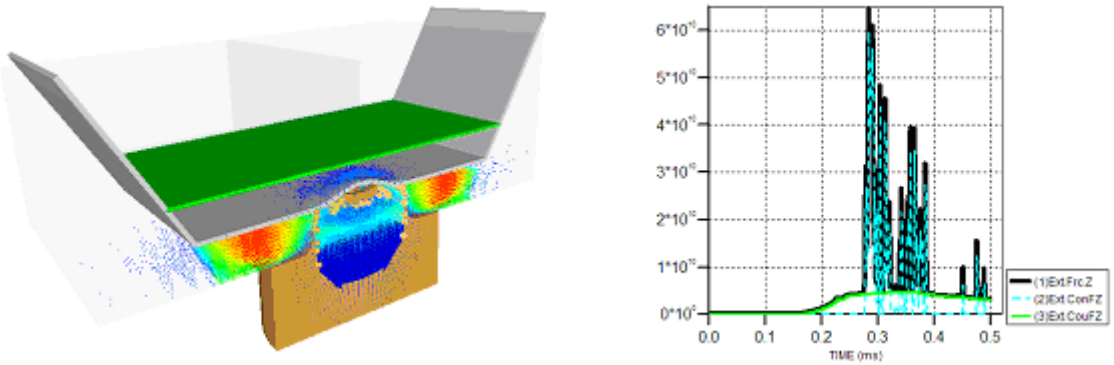
External forces

The resultant external forces on a Part can now be easily extracted from a simulation by selecting the appropriate components as Part summary variables. The components available are

- X/Y/Z contact forces
  External forces applied to a Part from Lag/Lag contact
- X/Y/Z coupling forces
  External forces applied to a Part from Euler/Lag coupling
- X/Y/Z boundary forces
  External forces applied to Part from all boundary conditions
- X/Y/Z total external forces
  The sum of all external forces acting on a Part

Note: External forces are not extracted for polygon based Euler Lagrange coupling in 2D

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

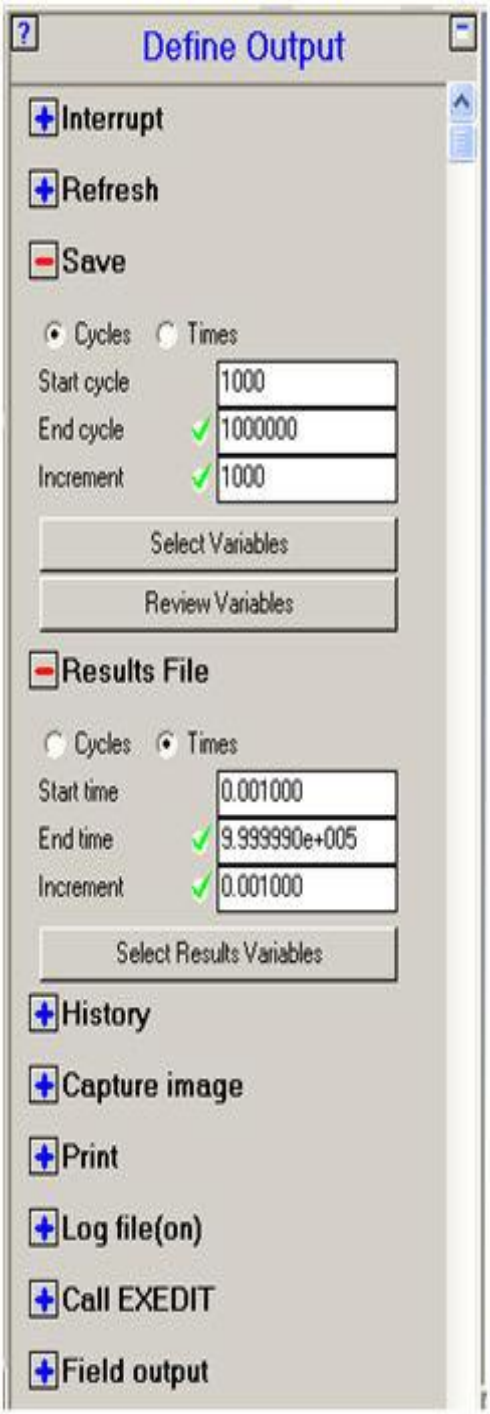


Example mine blast simulation highlighting source of loading (blast from coupling, impulse from soil ejecta) using new Part summary options

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Outputs-Results Files

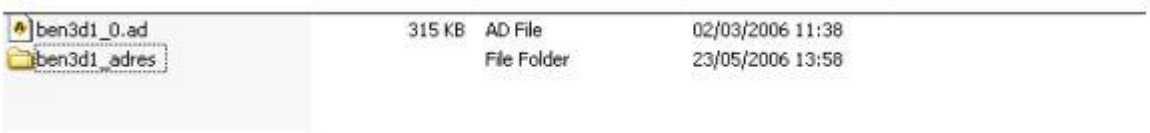
The capability to output results files, which contain a reduced amount of information, is available in AUTODYN-3D. This allows the post-processing of large simulations on computers with limited resources. Results Files cannot be used for restarting a problem. The frequency of the results files is independent of the SAVE files. The control for results file output is located in the Output panel, under Results Files:



From here, the frequency of results file output and the variable to save into the file are selected. Note that only a limited set of results variables are selected by default.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

When a simulation is performed with the results file option switched on, a new directory will be created in the directory where the simulation file is located. This directory will have the name <*ident*>*_adres*. For example, the results files from a simulation with model name ben3d1_0.ad will be stored in a sub-directory with name ben3d1_adres.



Results files have extension adres, and the model name is appended with two numbers:

- The first number is the cycle from which the simulation began.
- The second number being the current cycle number.

For example, a results file ben3d1_10_25.adres indicates that the results file contains data at cycle 25 from the simulation ben3d1 that began at cycle 10.

Note that the start cycle for each sequence of results files is stored in a file <*ident*>_<*cycle*>.ad_base. For example ben3d1_10.ad_base. This base file is essential for post processing using results files and must not be deleted. The base file contains data required for producing graphic output that for efficiency is not repeated in each of the Results Files.

To use the results files for post-processing of a simulation, the option under the File menu option should be

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

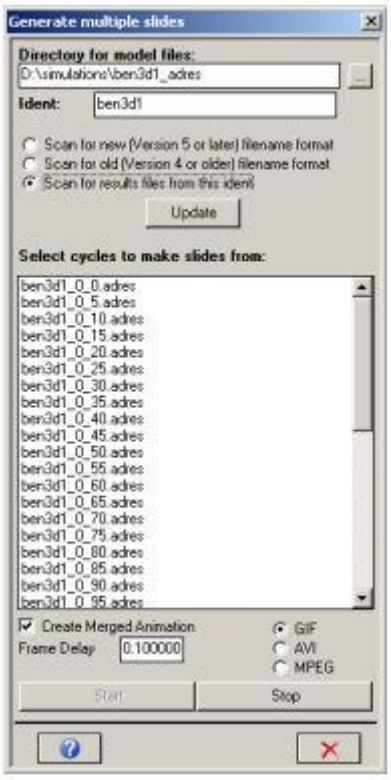
Once a results file has been selected, various options within AUTODYN will be inactive, since no model alteration is possible. The cycle drop-down menu from the Plots panel is automatically updated to represent the cycles of results files that can be loaded. In order to switch back to full restart files simply load in a restart (cycle) file.

The adres directory is designed to be independent of any other files, in order to share the results with colleagues simply transfer the directory.

Post-processing using results files is significantly more memory efficient than from a restart file. Using results files, AUTODYN only loads the data required to generate the requested Plots. The efficiency improvement will be problem dependant however a 2 to 3 times increase in the size of model that can be post-processed on a 32-bit Windows system can be expected.

To use the results files to create an animation of the simulation, select the option "Scan for results files from this ident" from the Generate Multiple Slides panel.

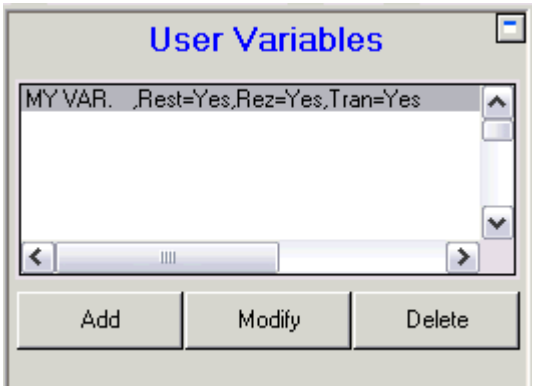
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information
of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

User Variables

User Variables



This panel lets you set up grid variables that you can use when programming your own user subroutines.

User Variable List

The top box lists the user variables you have currently defined. It shows the name you have assigned to the variable and whether or not the variable is written to restart (Save) files, rezoned and transported (in Euler). You can select a variable in this list.

Add

Press this button to add a new user variable.

Modify

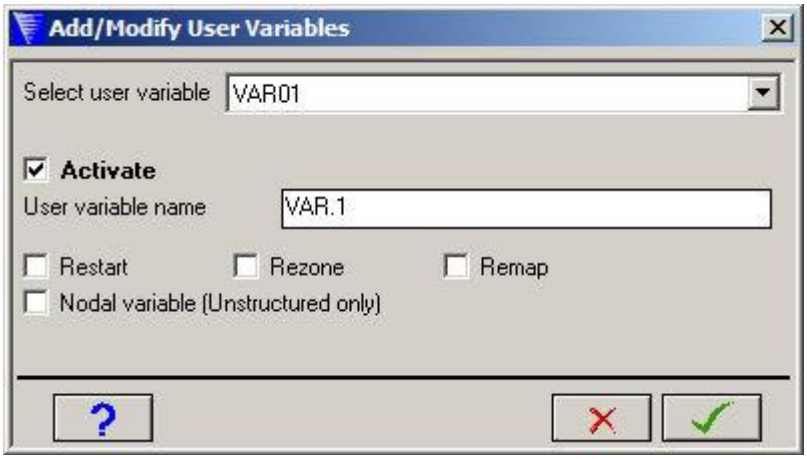
Press this button to modify the selected user variable.

Delete

Press this button to delete the selected user variable.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

User Variables - Add/Modify



This window lets you add a new user variable or modify the selected user variable.

Select user variable

You can define up to 20 user variables. Use this pull-down menu to select the user variable you want to define or modify.

Activate

Check this box to activate (define) the selected user variable.

User variable name

Enter the name you want to use for your user variable.

Restart

Check this box if you want the user variable to be written to the restart (Save) file.

Rezone

Check this box if you want the user variable to be included in the list of variables that are automatically rezoned.

Transport

Check this box if you want the user variable to be included in the list of variables that are automatically transported in Euler Parts.

Nodal Variable

For the unstructured solvers, element and nodal quantities are stored independently. Check this box to define a user variable that is associated with a node, rather than

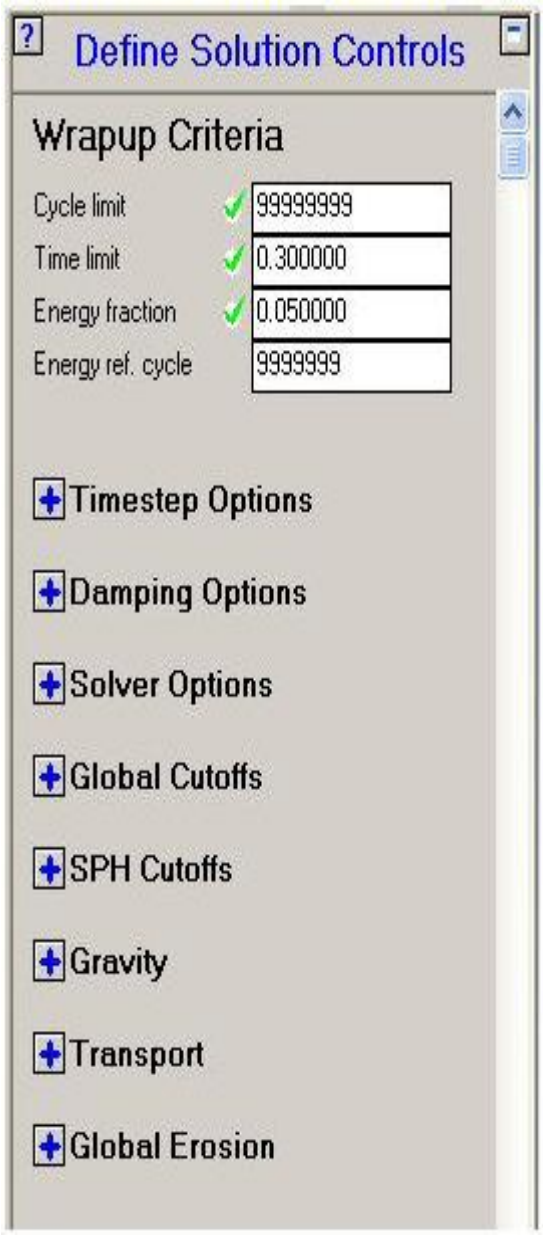
Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

an element for the unstructured solvers. Note that nodal user variable data can only be interrogated via examine and time history plots. Contour plots are not available.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Controls

Controls



This panel lets you define solution controls for your model.

Wrapup Criteria

These controls are the only ones visible when you open this panel for the first time. This is because you must specify some of the parameters.
(all other control inputs have default values that are usually OK).

Cycle limit

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Enter the maximum number of cycles you want your problem to run (enter a large integer if you don't want your problem to stop on a cycle limit).

Time limit

Enter the maximum time you want your problem to run (enter a large number - e.g. 1.0E20 - if you don't want your problem to stop on a time limit).

Energy fraction

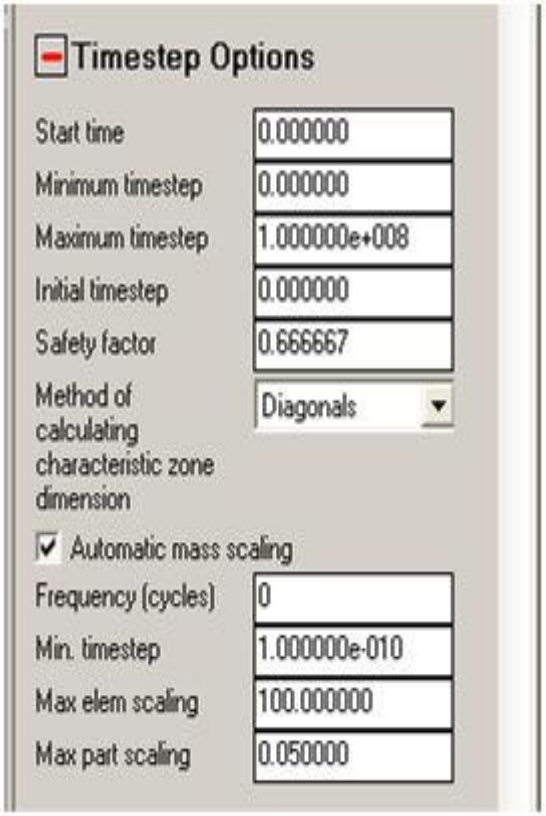
Enter the energy fraction you want to use to stop your problem if the energy error gets too large. The default value of 0.05 causes you problem to stop if the energy error exceeds 5%.

Energy reference cycle

Enter the cycle # at which you want AUTODYN to start checking for an energy error.

Press + alongside any of the remaining options to reveal the input fields for that option :

Timestep Options



These options control the timestep AUTODYN uses to run your problem.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Start time

Enter the start time for your problem.

Minimum timestep

Enter the minimum timestep you want to use.

If the timestep drops below this value, your problem will be terminated.

If you leave this field zero, the minimum timestep will be set to 1/10 of the Initial timestep.

Maximum timestep

Enter the maximum timestep you want to use.

AUTODYN will use the minimum of this value or the computed stability timestep.

Initial timestep

Enter the initial timestep you want to use.

If you leave this field zero, the initial timestep will be set to 1/2 the computed stability timestep.

Safety factor

It is not wise to run problem at the stability limit, so a safety factor is applied to the computed stability timestep.

Enter the stability factor you want to use.

The default value (0.6666) is good for all problems, but you may want to increase this value to 0.9 in some cases to use a larger timestep (0.9 is a good value for most Lagrangian calculations).

Method of calculating characteristics zone

Three options are available for calculating the effective zone size used in the timestep calculation. By default, all models will use a zone size calculation based on element diagonals. If the user wishes to obtain improved efficiency, especially in 3D simulations, the zone size calculation can be switched to one of two alternative options (based on either distances between opposing face centers or closest face centers).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

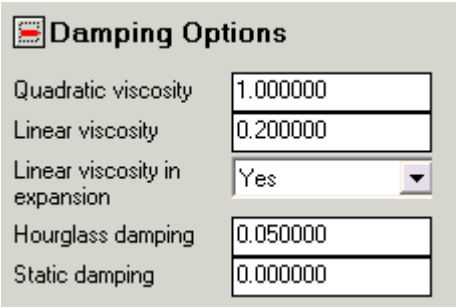
To obtain the optimal time step for HEX solid elements the method of calculating the characteristic zone dimension needs to be set to Opposing Faces option instead of the default Diagonal option.

Experience to date has shown that these options can significantly improve the efficiency of Lagrange simulations, especially in 3D. However, in certain circumstances when cells become highly distorted instabilities have been observed causing the calculation to terminate with high energy errors. The correct choice of erosion strain can reduce these problems. It is therefore recommended that users only utilize either of these non-default zone_size options if efficiency is critical.

Automatic Mass Scaling

Mass scaling is an artificial (numerical) mechanism for increasing the CFL (Courant-Friedrichs-Lewy) timestep of individual elements that govern the maximum allowed timestep of explicit transient dynamic solutions in AUTODYN. Increasing the timestep has the obvious benefit of reducing the number of cycles required to run a simulation to a given point in time. Educated use of this option can therefore result in significant improvements in efficiency.

Damping Options



These options control the magnitude of damping applied to your problem.

The default values are best for most problems.

Be very careful if you increase these values, they can significantly change your solution.

Quadratic viscosity

Enter the coefficient of quadratic viscosity you want to apply to shocks (compression waves).

Linear viscosity

Enter the coefficient of linear viscosity you want to apply to shocks (compression waves).

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Linear viscosity in expansion

Choose whether you also want to apply the linear viscosity in expansion.

Hourglass damping

Enter the coefficient of hourglass damping that you want to apply to Lagrangian elements.

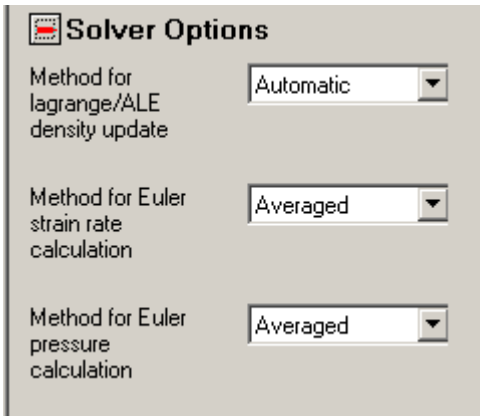
Static damping

A static damping constant may be specified which changes the AUTODYN calculation from a dynamic solution to a relaxation iteration converging to a state of stress equilibrium. For optimal convergence, the value chosen for the constant, R, may be defined by:

R=2*timestep/T

where timestep is the expected average value of the timestep and T is longest period of vibration for the system being analyzed.

Solver Options



Method for Lagrange/ALE density update

The element density can be updated each cycle using one of two methods; incrementally from the change in volume relative to the last cycle or recalculated based on the current mass and volume of the element. If the incremental change in volume over a cycle is small, it can be more accurate to use the incremental approach rather than the total approach to updating the density. This especially applies to the single precision version of AUTODYN.

Hence the options are:

Automatic - AUTODYN decides whether it needs to do an incremental update or not based on the change in volume.

Incremental - Force AUTODYN to always use the incremental update.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Total - Force AUTODYN to always recalculate the density.

Method for Euler Strain rate calculation

For multi-material Euler there are two methods to compute strain rates.

Weighted. For multi-material Euler analysis, involving materials with strength, strain rates need to be calculated to compute the increment in strain from cycle to cycle. The strain rates are derived from the velocity gradients, which are obtained by an element surface integration of the velocity. Using the Weighted option the face velocity that is used in the integration is a density weighted average of the element velocity on both sides of the face.

This is the default option and provides the most accurate strain rate calculation.

Averaged. With this option the face velocity used in the integration is a simple average of the element velocity on both sides of the face and provides less accurate results than the Weighted option. Until AUTODYN Version 6.1 the Averaged option was the only option available, and can be used to reproduce old results. Also models that are generated before Version 6.1 will still run using the Averaged method.

Method for Euler pressure calculation

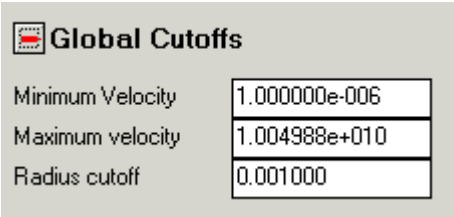
For multi-material Euler cells containing two or more materials, there are two methods for calculating the resultant cell pressure:

Equilibrium - An iterative procedure is used for each material in a cell to establish a single pressure consistent with the equation of state and conditions of each material in the cell. This method is currently only applied if there are two materials in the cell and these materials are defined by ideal gas, or fully burnt JWL material. If this criterion is not met in a cell the pressure averaging option is used.

Average - The resultant cell pressure is calculated from a weighted average of the pressure of the individual materials in the cell. The weighting algorithm takes account of both the volume fraction of each material and the relative stiffness of each material. This is the default method.

Global Cutoffs

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5



These options let you set global cutoffs.

The default values are usually good if you are using the default units (mm,mg,ms). You may want to change one or more of these defaults if you are using some other set of units.

Minimum Velocity

Enter the minimum velocity cutoff you want to use.

If any velocity drops below this value it is set to zero.

Maximum Velocity

Enter the maximum velocity you want to use.

If any velocity rises above this value it is set to this value.

This option is sometimes useful if your problem develops high velocities, which result in a small timestep, in regions that are not important to your solution.

Radius cutoff

Enter a value for the radius cuttoff for 2D axial models.

If a Lagrangian node is inside this radius at the start of your calculation, it is placed on the axis of symmetry.

If a Lagrangian node is outside this radius at the start of your calculation, it will not be allowed to come closer than this radius to the axis of symmetry as your calculation proceeds.

SPH Cutoffs

These options let you set SPH cutoffs.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

The default values are usually good if you are using the default units (mm,mg,ms). You may want to change one or more of these defaults if you are using some other set of units.

Limit density / Delete node

If the density of an SPH node drops below the minimum density or above the maximum density, you have the choice of either limiting the density to the cutoff density or deleting the node. Make your choice here.

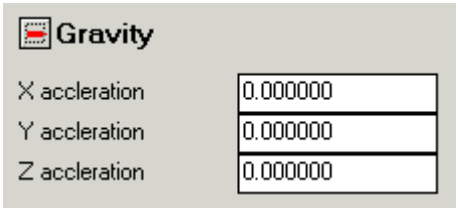
Minimum density

Enter the minimum cutoff density for SPH nodes.

Maximum density

Enter the maximum cutoff density for SPH nodes.

Gravity



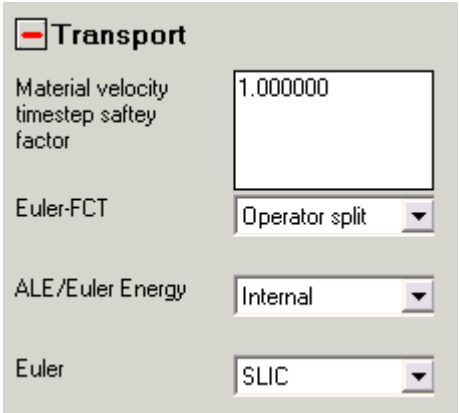
This option lets you set gravitational components for your model.

X / Y / Z acceleration

Enter the X Y and Z components of gravity in these fields.

The gravitational acceleration components will be applied to all Parts present in the model.

Transport



Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

These options relate to the transport of material in various Solvers.

Material velocity timestep safety factor

Enter the timestep safety factor for material velocities.

In the velocity used to calculate the stability timestep is **c+u**, where **c** is the soundspeed of the material and **u** is its physical velocity.

If **u** is of the order of or greater than **c**, then using **c + u/f** can give more stable results, where **f** is the safety factor.

Euler-FCT

Select the type of transport algorithm that is used for Euler-FCT calculations from this pull-down menu. The default setting is multi-dimensional where all three spatial dimensions are solved simultaneously which is more efficient and helps to maintain symmetric flow fields. The alternative is Operator Split where each spatial dimension is solved in turn.

ALE/Euler Energy

During a computational Euler cycle mass, momentum and energy is being transported (fluxed) from element to element. This option determines if total energy or internal energy is being transported.

Total. This option conserves the total energy (kinetic + internal energy) during transport. This is the most robust and accurate option and is the default.

Internal. This option conserves the internal energy during transport. In cases where the total energy transport doesn't provide an accurate answer, e.g. material overheating or when the kinetic energy is predominant, the Internal option might be chosen as an alternative.

Euler

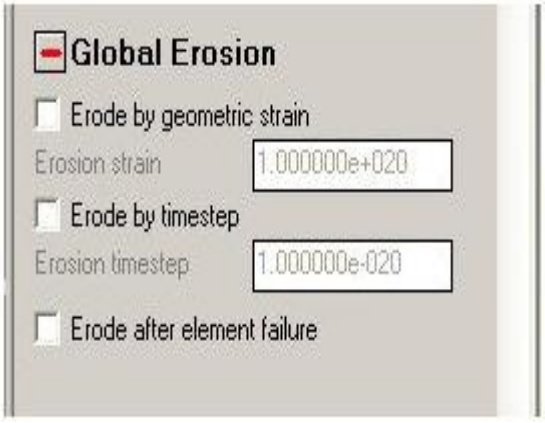
This option determines which transport algorithm will be used for multi-material transport.

SLIC. When material is being transported out of an element that contains multiple materials or voids, it is necessary to use special rules to determine the amount of each material being transported, because the only information available for the transport algorithm is the fraction of each material in the individual elements. In AUTODYN the Simple Lime Interface Calculation (SLIC) method is used which uses a small number of stereotypes of mixed material element topologies. The SLIC algorithm is the default and works well for all multi-material Euler analysis.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Updated SLIC. Isolated small parts of material smaller than an element do not move through the mesh at the correct velocity. They tend to stop moving while surrounding material rushes by. The Updated SLIC option will transport these small material fractions with statistically the correct velocity. This option should be used with care, because it can have some undesirable side effects. It is possible for these small material fragments to "tunnel" through other multi-material Euler objects like walls or plates and emerge at the other side.

Global Erosion



These options define global erosion criteria for all elements in the model. The global erosion options are only available for unstructured elements.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Controls-Mass scaling

Mass scaling is an artificial (numerical) mechanism for increasing the CFL (Courant-Friedrichs-Levy) timestep of individual elements that govern the maximum allowed timestep of explicit transient dynamic solutions in AUTODYN. Increasing the timestep has the obvious benefit of reducing the number of cycles required to run a simulation to a given point in time. Educated use of this option can therefore result in significant improvements in efficiency.

The primary use of mass scaling is to increase the CFL timestep of a small number of very small (or stiff) elements in the model, which otherwise would have controlled the timestep for all elements. Using this option, AUTODYN will automatically add artificial mass to individual elements to ensure that their CFL timestep is at least equal to a value defined by you. Mass scaling should only be used if the increased inertia of the scaled elements does not significantly affect the results of the simulation.

Mass scaling can also be used to increase the CFL timestep of elements that become highly distorted during the simulation. Extreme care should be taken to ensure that results obtained remain physical.

Mass scaling is activated under ***Controls, Timestep*** and the input parameters are defined as follows:

Automatic Mass Scaling	-	Activate automatic mass scaling
Frequency (cycles)	-	Cycle frequency that AUTODYN will adjust the element mass scaling. Recommended default is only at start-up (cycle 0). Post cycle 0 adjustment will only take place for solid elements.
Min. CFL timestep	-	Mass scaling is applied to all elements with a CFL timestep less than the specified value
Max. element scaling	-	Maximum mass scaling factor that can be applied to an element. Default is 100 (equating to 100 times original mass). If this limit is exceeded, no more mass is added to this element.
Max. part scaling	-	Maximum added mass that is allowed for an AUTODYN Part, as a fraction of the original Part mass. Default is 0.05 (equating to 5% of the original mass. If this value is exceeded, the simulation will terminate with an error message.

Two new element variables have been introduced with the development. These variables are automatically activated when mass scaling is selected:

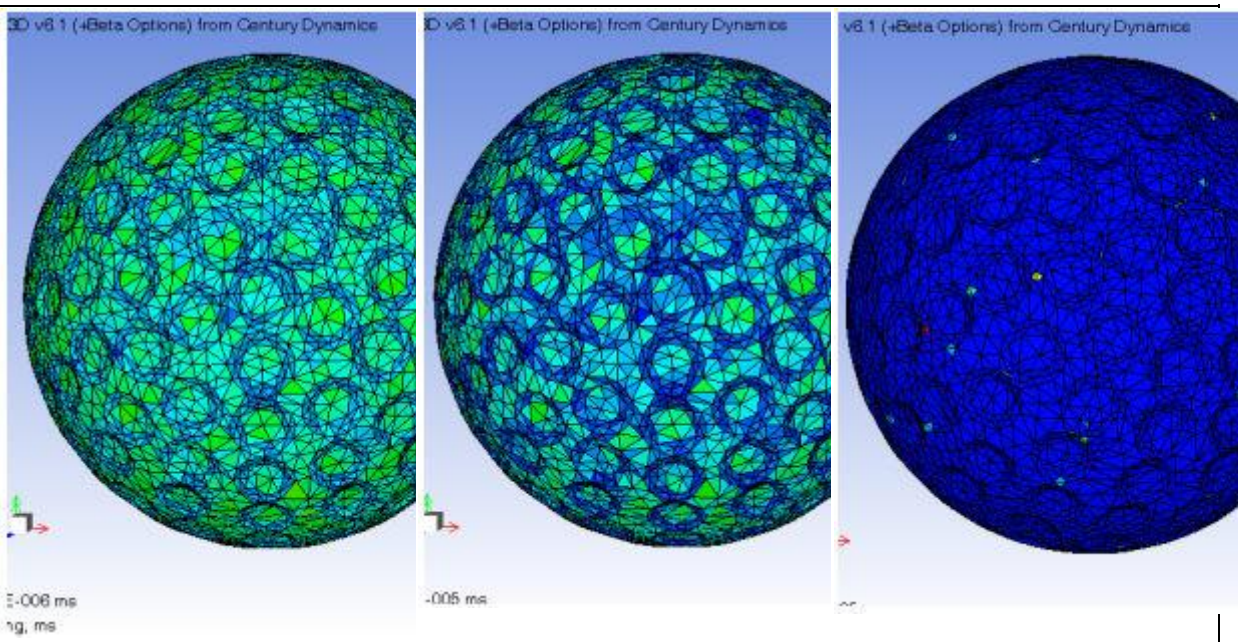
TIMESTEP: The CFL timestep for the element

MASS.SCALE: The mass scale factor applied to the element.
(1.0 indicates zero added mass, 2.0 indicates 100% added mass etc.)

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

A typical procedure to invoke mass scaling would be as follows

- Activate mass scaling retaining all the default parameters
- Run the simulation for 1 cycle
- Plot a contour of Timestep and review the element CFL timesteps. Identify a suitable timestep for mass scaling. The value chosen should exclude the majority of elements from mass scaling, and especially those in regions critical to the simulation results
- Re-load cycle zero
- Enter the selected Min CFL Timestep under Controls, Timestep
- Run the simulation



Minimum CFL timestep = 2.0e-5

Mass Scale CFL timestep = 1.0e-4
=>
Minimum CFL timestep = 1.0e-4

Small number of elements are mass scaled to maximum factor of 5.0

MASS SCALING SUMMARY BY PART

PART	MASS	ADDED MASS	TOTAL MASS	%ADDED
1-VOLUME	2.307E+04	4.945E+00	2.307E+04	2.144E-02
2-VOLUME	2.035E+05	0.000E+00	2.035E+05	0.000E+00

Timestep increased by factor of 5, for the addition of 0.02% of the Part mass

Example application of mass scaling

Important notes and limitations:

- Mass scaling will only be applied to Unstructured Elements/Parts
- Mass scaling is only applied to elements filled with materials which use a Linear or Hyperelastic equation of state
- The actual timestep used in a simulation will generally be lower than the Min CFL timestep you define

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

- A safety factor is applied to get the actual timestep (default = 2/3)
- The timestep used at cycle 1 of the simulation is additionally reduced by a factor of 2. The timestep is then allowed to increase by a maximum of 10% per cycle.
- Large velocities can affect the timestep at cycle 0
- Other features of the code will influence the timestep, such as artificial viscosity, blending, Gap contact.

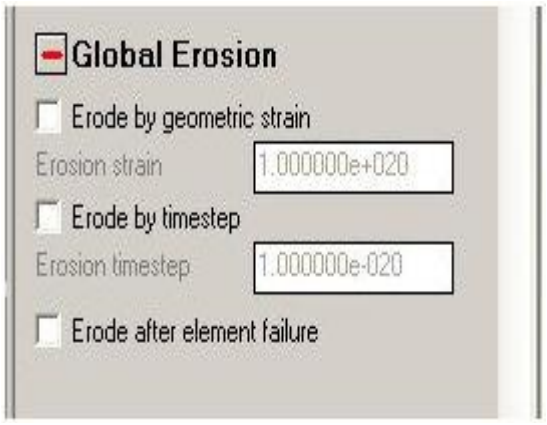
For Beams and Tet ANP elements, the minimum CFL timestep will be increased by using mass scaling. However, due to nodal averaging of some quantities that influence the timestep, the actual user defined CFL timestep may not be achieved.
A summary of the contribution of mass scaling applied to the model can be obtained through

- Plotting a contour of variable Mass Scale. This is a ratio of current mass to original mass (with scaling).
- Review the .prt file output activated using ***Output, Print, Energy Summary***

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5

Controls - Global Erosion

A new global erosion option has been implemented that can be used to define global erosion criteria for all elements in the model. The global erosion options are only available for unstructured elements.



Three erosion criteria have been implemented and can be used in combination in an analysis:

- Global Erosion by Geometric Strain
  - The user can define a single erosion strain that will be used for all elements in the model.
    - The minimum of the global erosion strain, and the local material erosion strain (if specified) will be used to check for element erosion.
- Global Erosion by Timestep
  - The user can define a single erosion timestep that will be used for all elements in the model.
    - The maximum of the global and the local material based erosion timestep (if specified) will be used to check for element erosion.
- Global Erosion by Failure
  - The user can turn on erosion by material failure for all materials.

Using erosion by timestep will activate the element timestep variable which can be used in post-processing.

Release 12.1 - © 2009 SAS IP, Inc. All rights reserved. – Contains proprietary and confidential information of ANSYS, Inc and its subsidiaries and affiliates.
Published: 2009-11-5